# CFD STUDIES ON FLOW THROUGH SCREW COMPRESSOR

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

Master of Technology

In

Mechanical Engineering

By

**G. Chanukya Reddy**

Department of Mechanical Engineering
National Institute of Technology
Rourkela

2007

# CFD STUDIES ON FLOW THROUGH SCREW COMPRESSOR

A THESIS SUBMITTED IN PARTIAL FULFILLMENT OF THE REQUIREMENTS FOR THE DEGREE OF

Master of Technology

In

Mechanical Engineering

By

**G. Chanukya Reddy**

Under the Guidance of

**Prof. Sunil Kr Sarangi.**

Department of Mechanical Engineering

National Institute of Technology

Rourkela

2007

**National Institute of Technology**

**Rourkela**

## CERTIFICATE

This is to certify that the thesis entitled, "CFD STUDIES ON FLOW THROUGH SCREW COMPRESSOR" submitted by Sri/Ms **G.Chanukya Reddy** in partial fulfillment of the requirements for the award of MASTER of Technology/ Bachelor of Technology Degree in **Mechanical Engineering** with specialization in "**Thermal Engineering**" at the National Institute of Technology, Rourkela (Deemed University) is an authentic work carried out by him/her under my/our supervision and guidance.

To the best of my/our knowledge, the matter embodied in the thesis has not been submitted to any other University/ Institute for the award of any degree or diploma.

Date:

Prof. Sunil Kr Sarangi
Dept.of Mechanical Engg.
National Institute of Technology
Rourkela - 769008

# ACKNOWLEDGEMENT

I extend my sincere gratitude and appreciation to the many people who helped keep me on track toward the completion of my thesis. Firstly, I owe the biggest thanks to my supervisor, **Prof.Sunil Kr Sarangi**, whose advice, patience, and care boosted my morale.

I am very much thankful to **Prof.R.K.Sahoo** and **A.K.Satapathy** , Lab in charge for CFD for their cooperation in completion of Thesis.

I am very much thankful to **Dr. N Sheshaiah,** who has been helped me a lot on screw compressor theory while doing his Ph.D .I am also thankful to **Mr.Anilkishan** for his valuable information on CFD package, and **Mr.Rajat bhuyan** for his valuable cooperation in fluent Lab.

I also thank all my friends, without whose support my life might have been miserable here. I wish to express my gratitude to my parents, whose love and encouragement have supported me throughout my education.

**G.Chanukya Reddy**

# Contents

# ABSTRACT

The twin-screw compressor is a positive displacement machine used for compressing gases to moderate pressures. It comprises of a pair of intermeshing rotors with helical grooves machined on them, contained in a casing which fits closely around them. The rotors and casing are separated by very small clearances. They may operate without internal lubrication, oil injected or with other fluids injected during the compression process. The rapid acceptance of screw compressors in various industries over the past thirty years is due to their relatively high rotational speeds compared to other types of positive displacement machines which make them compact, their ability to maintain high efficiencies over a wide range of operating pressures and flow rates and their long service life and high reliability.

Every time generation of different profiles and evaluate performance of those profiles by experiments is very difficult and these are expensive and time taking process. By using CFD can find out performance of different profiles easier.

The present work is aimed to create different screw compressor profiles in the GAMBIT. Generation of 3-D geometry and meshing of those profiles to evaluate performance of screw compressor in FLUENT has been done. Simulation of screw compressor working by using moving reference frame and dynamic mesh model in FLUENT is done.

Internal leakage, which has a significant impact on the efficiency and performance, is an inherent problem in the design of a twin-screw compressor. The objective of this study is to understand the leakage flow mechanisms, and quantify the leaking rate through each leakage pathways in the screw compressor. The numerical analysis is conducted using Computational Fluid Dynamics (CFD) software commercially available, FLUENT. The Realizable *k-e* turbulence model is employed because it has shown substantial improvements over the standard *k-e* model where the flow features include strong streamline curvature, vortices, and rotation. Results show that the highest individual leakage occurs across the male and female rotor tip sealing lines, and the blowhole.

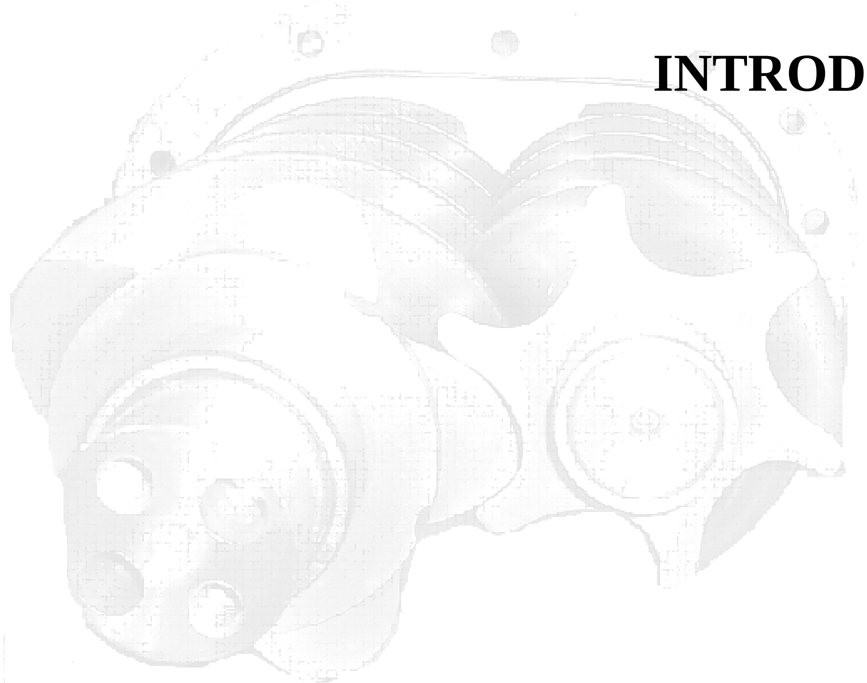
# List of Figures

**Chapter 5**

## LIST OF TABLES

# CHAPTER 1

## INTRODUCTION



Classification of Compressors

Twin-Screw Compressor Applications and Advantages

Geometry of Screw Compressor

Design Parameters

Performance Parameters

Objectives of the Investigation

Organization of the Thesis

# INTRODUCTION

Gas compressors are mechanical devices used for raising the pressure of gas or vapour either by lowering its volume (as in the case of positive displacement machines) or by imparting to it a high kinetic energy which is converted into pressure in a diffuser (as in the case of centrifugal machines). The classification and use of compressors are described in the next section.

The selection of compressors for different applications is a crucial issue in the process industry. It is usually the most expensive piece of equipment and has dominant influence on cycle efficiency. The common types of compressors used in industry are reciprocating, twin screw, single screw, centrifugal, scroll and rotary vane. Compressor manufacturers are used to having a large market potential. Probably all types of compressors can be improved over what is available in the market today but the potential return must justify the expense of research and development to achieve the improvement.

Screw compressors of the type that is employed in the process and gas industries are large and expensive, while their continuing function is usually essential for continuation of the entire process in which they play a part. The reliability of their operation is thus at least as important as their efficiency. In the past few years, significant advances have been made in the design and manufacture of the main components of machines of this type, such as the rotors and the bearings, as well as lesser components. These have resulted in previously unthinkable improvements to both performance and reliability that has been widely applied to both air and refrigeration compressors. Despite this, process gas compressors have not yet widely benefited from these advances because, for these applications, the numbers produced are far fewer and they have a far longer development cycle than other compressors. Thus, improvements in process gas compressors are now overdue. Moreover, experience already gained in other applications can be incorporated at the design stage, with the minimum of added development time and cost, by simultaneous consideration of all the relevant variables that affect their operation.

### Reliability A vital Demand in Gas and process Screw compressors

Screw compressors have a growing role in the gas and process industry, where their power requirement is high and machine sizes are large. They are simple machines in which the movement of the parts is purely rotational. Therefore, they are typically up to five times lighter than their reciprocating counterparts of the same capacity and have a nearly ten times longer operating life between overhauls.

Moreover, provided that the running clearances between the rotors and between the rotors and their housing are small, they can maintain high volumetric and adiabatic efficiencies over a wide range of operating pressures and flows. Specialised machine tools now enable the most complex screw rotor shapes to be manufactured with high tolerances at an affordable cost. Their use in screw compressor manufacture, together with advanced rolling element bearings, has led to substantial improvements in performance and reliability. The development of rotor profiles has recently been enhanced by advances in mathematical modelling and the computer simulation of the thermodynamic and fluid flow processes within the compressor. These analytical methods may be combined to form powerful tools for process analysis and optimisation and are steadily gaining in credibility as a means of improving design procedures. As a result, the design of screw compressors has evolved substantially over the past 10years and is likely to lead to additional improvements in machine performance in the near future.

## 1.1 CLASSIFICATION OF COMPRESSORS

Compressors are broadly divided into positive displacement and dynamic type. The detailed classification of compressors is shown in Figure 1.1. The dynamic principle is utilized in the multi blade dynamic compressors. These are further sub divided into centrifugal and axial flow types. In axial compressors, the velocity of fluid can be supersonic which is converted to a pressure head by diffusers. Positive displacement compressors are further subdivided into piston compressors in which the gas volume changes due to the action of one or more reciprocating pistons moving axially in the cylinder and Membrane compressors in which the volume variations are affected by deflection of an elastic partition. In the positive displacement rotary compressors, the compression is produced as the result of reduction of volume inside the rotating elements such as helical screws.

**Compressor Specifications:**

Most of the compressors are specified with the following parameters:

- Mass Flow Capacity.
- Inlet / Suction Pressure.
- Discharge / Operating Pressure.
- Inlet Temperature.

- Speed.
- Type and Volume of Gas Handled.

**Dynamic compressors**

Dynamic compressors are sub divided into radial flow (centrifugal) and axial flow compressors.

***(a) Centrifugal Compressor***

It consists of a vane rotating disk or impeller which is used to force the gas up to the rim whereby the speed of the gas is increased. The turning of the impeller causes compression of the gas. The compressor also incorporates a diffuser as the converting element from velocity to pressure head. Centrifugal compressors are usually stationary compressors. They are used for heavy duty applications in industry. They are most suitable to applications where the pressure requirement is moderate. They are widely used in large snow-making works especially in ski resorts and also in gas turbine engines. They are also used as turbochargers and superchargers in internal combustion engines. The large air separation industry also uses centrifugal compressors. The advantages of the centrifugal compressor are its ruggedness and relatively low cost of manufacturing. The disadvantages are its large frontal area and its relatively poor pressure ratio when compared with other types of compressors.

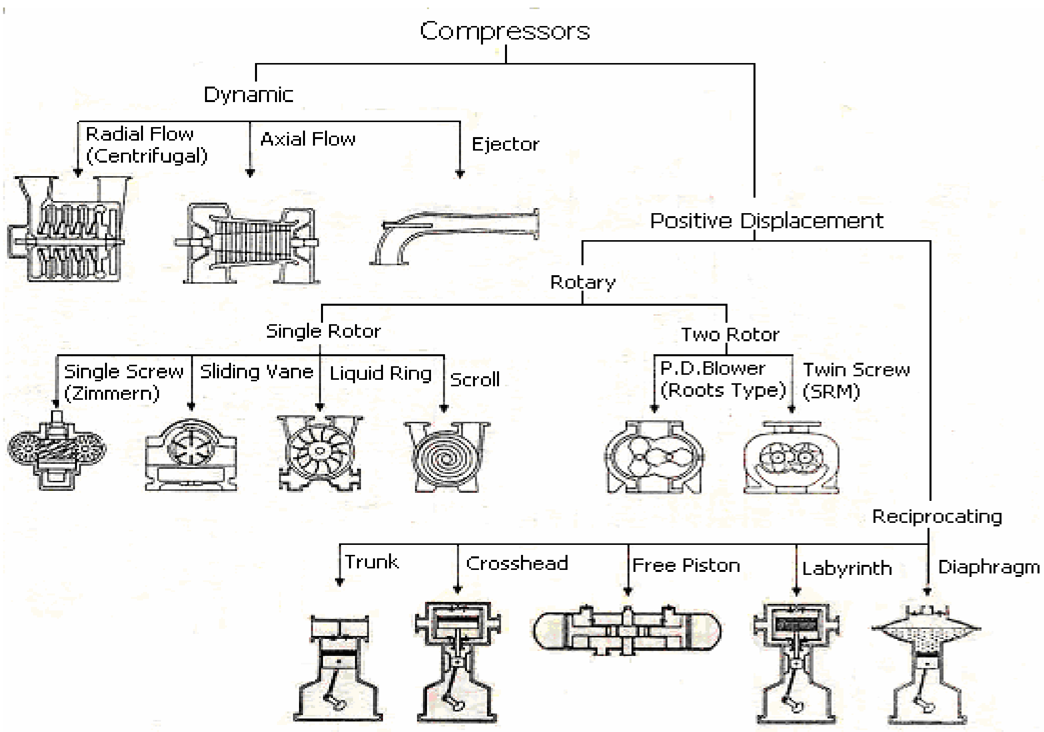



*Figure: 1.1: Classification of compressors [1].*

***(b) Diagonal or Mixed-Flow Compressor***

A mixed flow is similar in construction to that of the centrifugal compressor. It differs in that the axial and radial components are available outside the rotor. It incorporates a diffuser for turning the diagonal flow into the axial direction.

***(c) Axial Flow Compressor***

The axial flow compressor consists of rows of alternate rotating (rotor) and fixed (stator) blades. A row of rotors followed by a row of stators is known as a stage and there may be many stages attached to a single shaft. Gas is drawn usually by the spinning of the numerous fans on the shaft. A series of divergent and convergent ducts form the stator. Axial flow compressors are mostly available as multi staged type. They are used in applications that require very high flow such as large gas turbine engines.

**Positive Displacement Compressors**

In positive displacement compressors, the compression is realized by displacement of a solid boundary and preventing the fluid by this solid boundary from flowing back in the direction of pressure gradient. These are further classified into reciprocating and rotary types.

***(a) Reciprocating compressor***

As the piston moves for the downward stroke, discharge valves are forced shut. The gas is then sucked into the cylinder by the suction pressure. During upward stroke the suction valves are forced shut and the gas is moved into the discharge valve. The spring loaded valves are self operated by the pressure in the cylinder.

**Advantages**

- High efficiency, particularly when new and after overhauls.
- Possibility of very high discharge pressures.

**Disadvantages**

- Requires high maintenance.
- Fixed speed, Creates noisy atmosphere, Damping requirement to arrest vibration
- More moving parts, efficiency drops off between overhauls.
- High cost.

**Applications**

There are no rigid rules for the application of reciprocating compressors. The power ranges between 5 and 30 HP are used for automotive work especially for intermittent duties. Compressors of greater power such as 1000 HP are meant for large process applications.

### *(b) Rotary Compressors*

The compression is produced as a result of the positive action of rotating elements such as helical screws, scrolls or vanes. Rotary positive displacement compressors are classified into single rotor and two rotor compressors. Single rotor compressors are further classified into single screw, sliding vane/rotary vane, liquid ring, and scroll compressors. Similarly, the two rotor compressors are divided into roots blower and twin screw compressors.

#### (I) Single Rotor Compressor

The single rotor compressors are further classified as below.

##### *(i) Single-screw compressor*

This type of compressor uses a single main screw rotor meshing with two gate rotors with matching teeth. The main screw is driven by the prime mover, typically an electric motor. The gate rotors may be made of metal or a composite material. The screw-like grooves gather gas/vapours from the intake port, trap them in the pockets between the grooves and compressor housing, and force them to the discharge port along the meshing path. This action raises the trapped gas pressure to the discharge pressure. Single screw compressors usually employ hermetic or semi-hermetic designs for higher efficiency, minimum leakage and ease of service.

##### *(ii) Rotary Vane Compressor*

It consists of a single rotor accommodated in a cylindrical housing. As the rotor rotates, the vanes of the rotor are thrown off against the housing wall. The bearings and vanes are lubricated with oil. The segments created by the vanes change in shape and volume during a cycle. This results in the compression of the trapped gas. Ports are used for intake and exhaust of the gas at the minimum and maximum pressure positions.

*Advantages*

- ❖ Simplicity in design, slow rotational speed results in low wearing of parts.
- ❖ Moving parts are only the rotor and the vanes.
- ❖ The rotor can be directly driven; Gas discharge rate is not pulsed.
- ❖ No additional foundation required Vibration free operation.
- ❖ Valve-less operation, requires little maintenance.

*Disadvantages*

- ❖ Close contact between the compression gas and lubricating oil.
- ❖ Lubrication oil necessitates disposal.

- ❖ Low pressure capability

*Applications*

- ❖ Landfill gas gathering and boosting Digester mixing.
- ❖ Fuel gas boosting, Flare gas recovery.
- ❖ Wellhead gas compression.

***(iii) Liquid ring compressor***

In a liquid ring compressor the rotor is positioned centrally in an oval-shaped housing. Upon rotation, which proceeds without metal to metal contact, a ring of liquid is formed that moves with the rotor and follows the shape of the housing. At the two points of the closest proximity between the rotor and the housing, the liquid completely fills the chambers of the rotor. As rotation proceeds, it follows the contour of the body and recedes again, leaving spaces to be filled by the incoming gas. These spaces are connected via the cone porting to the inlet of the compressor. As a result of the suction action thus created, gas is pulled into the compressor. As the rotation progresses, the liquid is forced back into the chambers, compressing the gas. This gas is forced out of the discharge port and then leaves the compressor via the outlet flange.

***(iv) Scroll compressor***

The scroll compressor consists of two scroll members each with a spiral shaped wrap and corresponding end plate. The two scroll members are placed at the top of the compressor housing. The top scroll member is fixed to the compressor housing while the lower member is free to orbit about the centre of the fixed scroll. The moving scroll is driven by a motor and is attached to the motor shaft. As the moving scroll orbits about the fixed scroll, the wraps of the two scroll members form lines of contact. These lines of contact form crescent shaped symmetric pockets. The gas enters the first pair of symmetric pockets at the outer periphery of the scroll set and continues to travel towards the centre. As the gas moves towards the centre, the volume of the pockets decreases thus compressing the trapped gas.

Once the gas reaches the centre of the scroll set, the tip of the moving scroll begins to uncover the discharge port located in the centre of the endplate of the fixed scroll member. Once the discharge port becomes uncovered, the discharge process begins. Because of the scroll’s geometry, no discharge valve is needed; instead the tip of the moving scroll wrap uncovers the discharge port. Scroll compressors are used refrigeration and air conditioning systems as well as supercharger in automotive operations. Pulsed output is obtained which discourages its usage in industry.

***(v) Roots Blower***

It is a positive displacement compressor with two lobed impellers, each resembling the figure of "8". The two rotors are mechanically linked via gears such that they rotate in opposite directions. The figure of "8" shape allows the impellers to be close to but never in contact with each other and the compressor walls at every position of their rotation. The close tolerances of the components allow the impellers to move without a lubricant if required.

Roots blowers are typically used in applications where a large volume of gas must be moved across a relatively small pressure differential. This includes low vacuum applications, with the roots blower acting alone, or use as part of a high vacuum system in combination with other compressors. Roots blowers are also used as superchargers.

**(II) Twin Screw Compressor**

It is a positive displacement machine that uses a pair of intermeshing rotors housed in a suitable casing instead of piston to produce compression. In the double screw compressor, each rotor comprises of a set of helical lobes affixed to a shaft. One rotor is called the male rotor and the other rotor is the female rotor. The number of lobes on the male rotor, and the number of flutes on the female, will vary from one compressor manufacturer to another. However, the female rotor will always have numerically more valleys (flutes) than the male rotor lobes for better efficiency. Either an electric motor or an engine drives the male rotor.

Twin-screw compressors are basically classified into oil free and oil injection types. Single stage oil free machines are used for low-pressure devices and oil-injected compressors are used for moderate pressure machines. Multistage designs are used for the compressors working with higher pressure ratio. Oil injection provides cooling, lubrication and sealing, thus permitting higher pressure ratio.

***(i) Oil free compressor***

Oil free operation implies that the gas compression space is entirely free from oil contamination. These compressors are suitable for compression of a wide variety of gases requiring relatively low pressure ratios at fairly constant volume flow rates. These are particularly suitable for handling contaminated, particle-laden, polymerizing, or explosive gases.

### *(ii) Oil Injected compressor*

In an oil injected twin-screw compressor, the lubricating oil is deliberately injected into the gas stream to absorb the heat of compression. This enables much higher pressure ratio in a single stage without intercooling and provides significant protection against corrosive gases. Also, it permits the use of step less capacity control by slide valve. No timing gears are required and the male rotor usually drives the female rotor through an oil film between the intermeshing profiles.

Twin-screw oil injected machines are used today for different applications both as compressor and expander. They operate on a variety of working fluids. The working fluid may be gas, dry vapour, or multi phase mixture with phase change taking place within the machine. Presently, screw compressors hold an important position in the industrial refrigeration and compression field. It occupies a position alongside reciprocating and centrifugal types as a standard choice for refrigeration compressors.

The asymmetric rotor profile of oil-injected type screw compressors reduces the leakage path cross section resulting in increased efficiency. The helical screw compressor is the only rotary unit that operates at tip speeds in excess of 0.12 Mach [2]. These machines develop pressure 7.3 times more than that of a centrifugal compressor operating at the same tip speed [3]. While the performance characteristics and flexibilities of this type machine approach that of the piston compressor, it has compression efficiency comparable to that of a centrifugal machine. Its maintenance cost also compares favourably with that of the centrifugal machines, which is less than one third of the cost of maintaining piston compressors. A schematic view of a pair of twin-screw rotors is shown in Figure1.3.

The commercial range of oil injected screw compressors available today covers outer diameter of the male rotor between 75 and 620 mm. They produce between 10m$^3$/hr and 10,000 m$^3$/hr of gas flow rate for refrigeration and air conditioning applications and up to 60,000m$^3$/hr for general purpose applications. Pressure ratios of 3.5 for dry compressors and up to 15 for oil injected twin-screw compressors are available in the market. The average pressure difference is up to 15 bar but maximum pressure difference some times exceeds 40 bar [1]. Typically, the volumetric efficiency of oil injected compressor exceeds 90%.

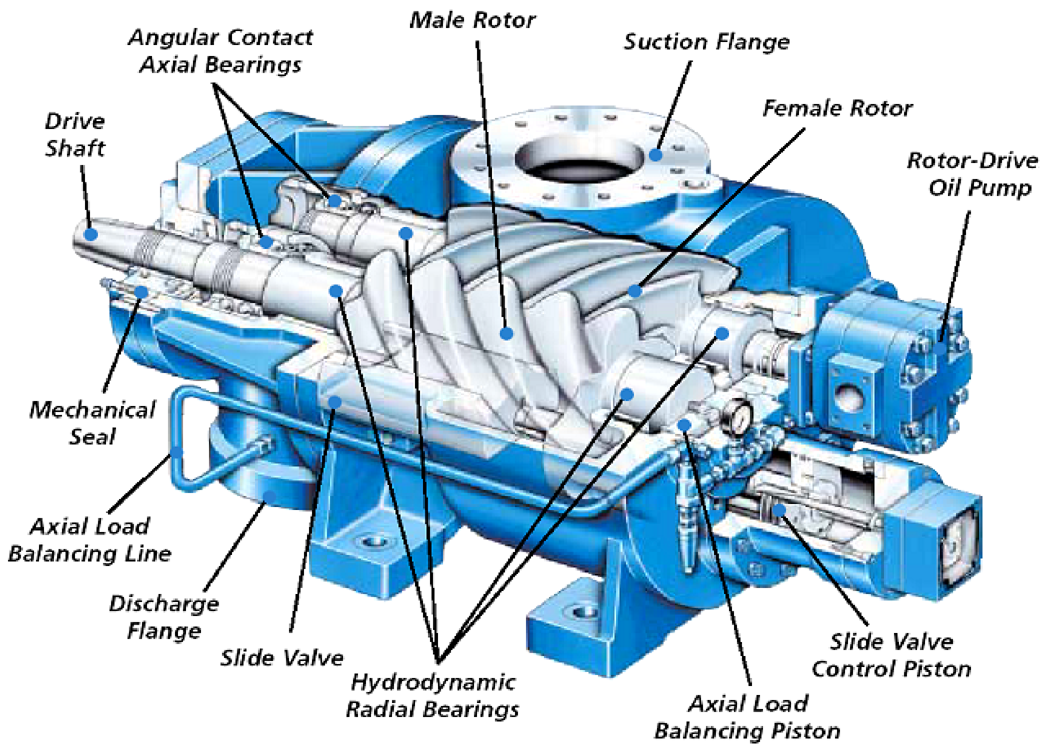



*Figure 1.2: Isometric cutaway drawing of Twin-Screw compressor [4]*

## 1.2 TWIN-SCREW COMPRESSOR APPLICATIONS AND ADVANTAGES

Oil injected twin-screw compressor offers several advantages over comparable machines. The advantages are:

(i) Simple maintenance, low maintenance cost and Low weight.

(ii) High reliability and long compressor life, full use of drive horsepower.

(iii) Low operating expense, medium purchase price.

(iv) High compression ratio (up to 16 per stage).

(v) Operation at low suction pressure up to 66 cm Hg vacuum.

(vi) Reduced package cost due to compactness.

(vii) Absence of reciprocating components and low internal forces, allowing the compressor to run at higher tip speeds, resulting in more compact unit.

(viii) The continuous flow of cooling lubricant permits high single stage compression ratio.

(ix) High speeds and compression ratios help to maximize the available power efficiency.

(x) Less sensitive to liquid slugging.

(xi) Suitable for wide variety of gases.

(xii) High ride quality and controllability.

(xiii) High speed operation over a wide range of operating pressures and flow rates with high efficiencies.

**Drawbacks**

(i) High volume of lubricant injected into a conventional screw compressor reduces power efficiency due to the oil compression phenomenon and increased stirring loss due to the presence of oil.

(ii) Power consumption during unloading operation is normally higher than that of reciprocating type.

(iii) Issues such as rotor deflection, casing strength, intrinsic leakage and technological deficiencies.

**Applications**

The rotary dual screw compressor package is ideal for numerous gas compression applications including fuel gas boosting, general construction and road building purposes and operating pneumatic tools. It has become a better choice in textile, electronic and automotive sectors, iron and steel, food and beverage, tobacco sector and petroleum refining and petrochemicals, vapour recovery, land fill and digester gas compression and propane/butane refrigeration compression. It can be applied for compression of corrosive and/or dirty process gas, poly-alpha-olefins, poly-glycols, synthesized hydrocarbons and liquefaction of gases.

A rotary compressor package can also be used to upgrade existing reciprocating compressor installations. By boosting low suction pressure, capacity may be increased at minimum cost with continued use of existing reciprocating equipment. If an application requires large volume flow rate at low suction pressure, but discharge pressures greater than what the screw machine can provide, a combination of screw and reciprocating units with a common driver can be a better solution.

**Materials**

Oil injected screw compressors are built with different materials for its components.

***(i) Casing***

Gray cast iron is generally used to manufacture casing for air and inert gases. Cast S.G iron is used for compression of hydrocarbon and other hazardous gases, where site conditions/specifications permit a less expensive material than steel. Cast carbon steel is necessary for clean hydrocarbon and other hazardous gases and even sour and/or wet gases.

***(ii) Rotors***

The rotors are normally made of ferrous materials. Aluminium and plastics are feasible in certain applications. Forged carbon steel is used for air and most gas applications. In many cases, S.G (nodular) iron is also acceptable. Forged high alloy steel is preferred (typically 12-13% Cr, 4-5% Ni) for corrosive, sour and wet gases.

## 1.3 GEOMETRY OF SCREW COMPRESSOR

The rotors of a twin-screw compressor are a form of helical gears with parallel axes and a uniform lead. The rotors make line contact and the meshing criterion in the transverse plane perpendicular to their axes is the same as that of spur gears. Two intermeshing rotors are housed in a suitable casing to achieve compression. Different views of compressor rotors and casing are shown in Figure 1.3 for better understanding of the geometry.

The rotors intermesh in the dual bores of a one-piece cylinder. The space between any two successive lobes of each rotor and its surrounding casing forms a separate working chamber of fixed cross sectional area. The length of working chamber varies as rotation proceeds due to displacement of the line contact between the two rotors. The volume is maximum when the entire length between the lobes is unobstructed by the meshing contact between the rotors and has a minimum value of zero when there is full meshing contact with the second rotor at the discharge end. The two meshing rotors effectively form a pair of helical gear wheels with their lobes acting as teeth. The casing provides gas inlet passages, oil injection points, and compression zone and discharge ports. Rolling element bearings located near the ends of the rotor body support the screws.

Majority of screw compressors are still manufactured with four lobes in the main (male) rotor and six lobes in the gate (female) rotor with the same outer diameter. This configuration is a compromise having favourable features for both dry and oil injected compressor applications and is used for air and refrigeration or process gas compression. However, other configurations like 5/6 and 5/7, and more recently 4/5 and 3/5, are becoming increasingly popular. A configuration with five lobes in the main rotor is suitable for higher pressure ratios, especially when combined with larger helix angles. The 4/5 arrangement has emerged as the best combination for oil injected applications with moderate pressure ratios. The 3/5 configuration is favoured in dry applications, because it offers a high gear ratio between the gate and the main rotors, which may be taken advantage of to reduce the required drive shaft speed.

However, the efficient operation of screw compressors is mainly dependent on proper rotor design. Most commonly used rotor profiles are shown in Figure 1.5. An additional and important requirement for successful design of all types of compressors is an ability to predict accurately the effects of the change in any design parameter on performance. The main requirement of these types of compressors is to improve the rotor profile so that the flow area through the compressor is maximized while the internal leakage areas are minimized. Also, the internal friction due to relative motion between the contacting rotor surfaces should be made as small as possible.

Compressor designs have evolved gradually over half a century and the present trend is to realise smallest possible machines still meeting the required performance. This means, the rotor tip speeds are to be as high as possible with the limits imposed by efficiency requirements. Rolling element bearings are to be used to permit the smaller clearances than journal bearings. Similarly the ports are to be made as large as possible to minimize suction and discharge gas speeds, and consequent pressure losses.

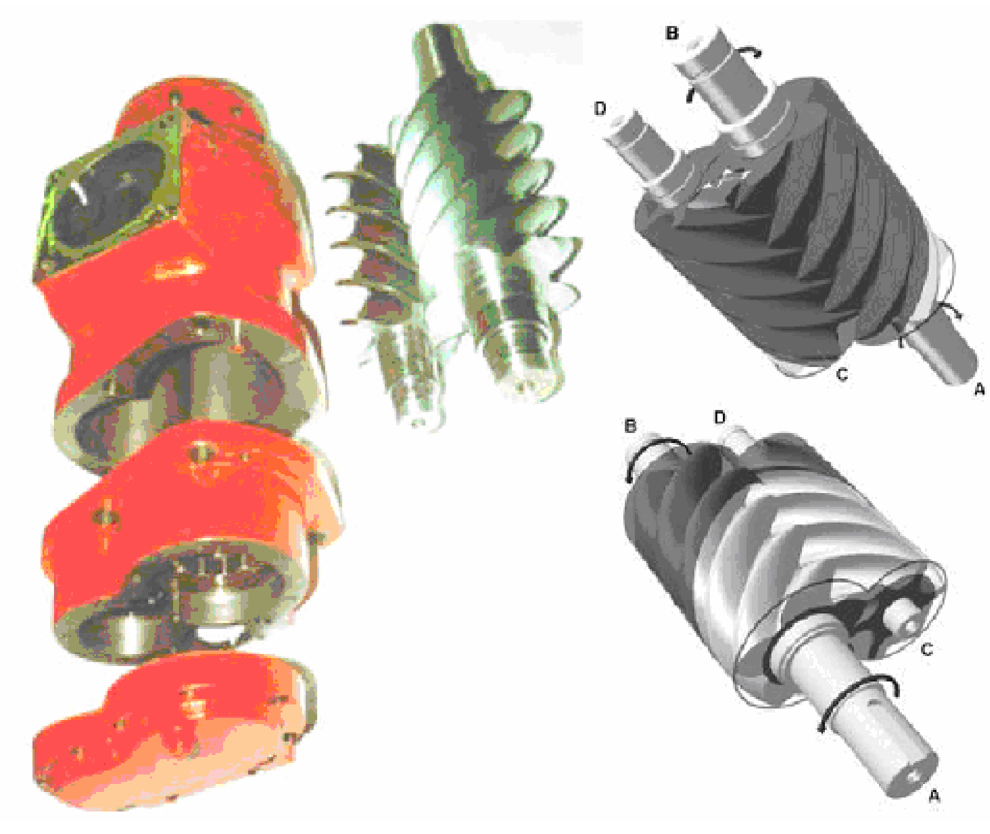



*Figure 1.3: Different views of meshing screw rotors and casing [5]*

An efficient screw compressor needs a rotor profile which has a large flow cross-section area, short sealing line and small blowhole area. The larger the cross sectional area, the higher is the flow rate for the same rotor size and rotor speed. Short sealing lines and smaller blowhole area reduce leakages. Higher flow rates and smaller leakage rates increase compressor volumetric efficiency. This, in turn, increases the adiabatic efficiency because less power is wasted to compress internally re circulated leakage gas.

Precision manufacture permits rotor clearances to be reduced, but the likelihood of direct rotor contact is increased despite oil flooding. Hard rotor contact leads to the deformation of the female rotor due to increased contact forces, and ultimately leads to rotor seizure. Hence, the screw profile should be designed in such a manner that the risk of seizure is minimized. The clearance between the rotor and the housing, especially at the high-pressure end, must be properly selected. This in turn requires either expensive bearings with smaller clearances or cheaper bearings with their clearances reduced to an acceptable value by preloading.

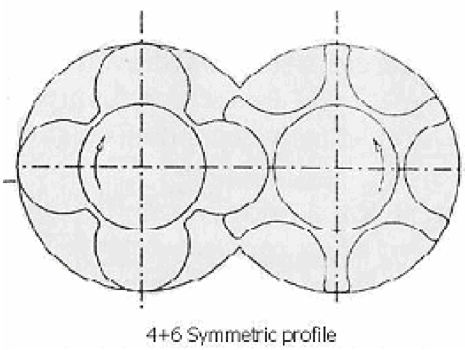
Oil injected screw compressor which operates with high pressure difference is heavily loaded by axial and radial forces, which are transferred to the housing by the bearings. Rolling element bearings are normally chosen for small and medium screw compressors and these must be carefully selected to obtain a satisfactory design. Usually two bearings are employed on the discharge end of the rotor shafts in order to absorb the radial and axial loads separately. The contact force between the rotors is determined by the torque transferred between them and is significant when the rotors make direct contact. It is relatively small when the compressor drive is through the main rotor. If the drive is through the gate rotor, the contact forces will be substantially larger and as far as possible, this arrangement should be avoided.

The oil used for sealing the gaps is also used for bearing lubrication. Generally the oil supply to the bearings is separate to minimize frictional losses. Oil is injected into the compressor chamber at the place where thermodynamic calculations show the gas and oil inlet temperatures to coincide. The position is defined on the rotor helix for locating the injection hole located so that the oil enters tangentially in line with the female rotor tip to recover the maximum possible oil kinetic energy.

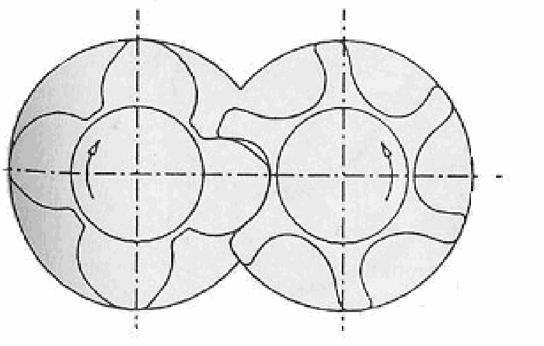
To minimize the flow losses in the suction and discharge ports, the suction port is positioned in the housing in a manner that lets the gas enter with the fewest possible bends and the gas approach velocity is kept low by making the flow area as large as possible. The discharge port size is determined by estimating the built-in-volume ratio required for optimum thermodynamic performance. The discharge port position is so adjusted as to reduce the exit gas velocity to minimum to obtain the lowest internal and discharge flow losses. The casing should be carefully dimensioned to minimize its weight containing reinforcing bars across the suction port to improve rigidity at higher pressures.

## 1.4 DESIGN PARAMETERS

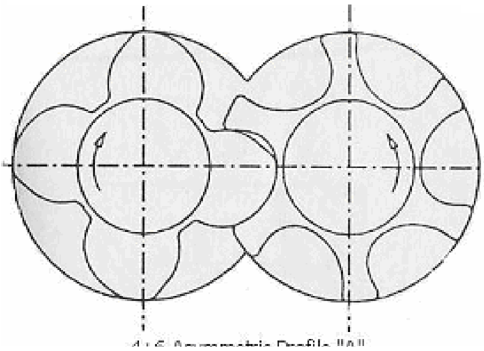
The performance of a screw compressor depends on a large number of design parameters. Knowledge of the effect of these parameters can help a designer select the best performing machine for a given application. It can be improved considerably by proper selection of some of the important design parameters which influence the performance.



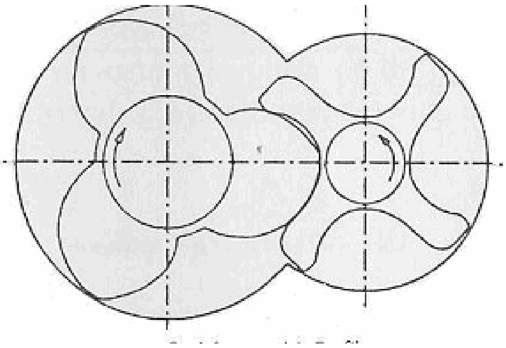

*4/6 Symmetric profile*
*Predominantly used for oil free compressors*



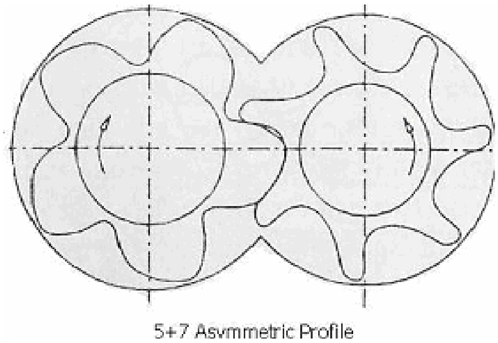
*4/6 Asymmetric profile "D"*
*Predominantly used for small oil injected*



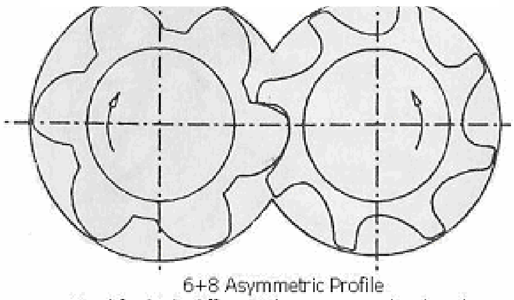
*4/6 Asymmetric profile "A"*
*Predominantly used for larger oil injected compressors*



*3/4 Asymmetric profile*
*Used for high volume flow, low differential pressure applications*




*5/7 Asymmetric profile*
*Used in place of 4/6 for smaller rotor diameters to improve shaft rigidity*




*6/8 Asymmetric profile*
*Used for high differential pressure and reduced volume flow applications*

**Cycle of operation**

The compression process in a screw compressor is similar to that of a reciprocating compressor, but it does not suffer re-expansion at the end of compression cycle from clearance volume. Its p-v diagram is, therefore, very simple and it is similar to the reciprocating compressor without clearance volume as shown in Figure 1.5.

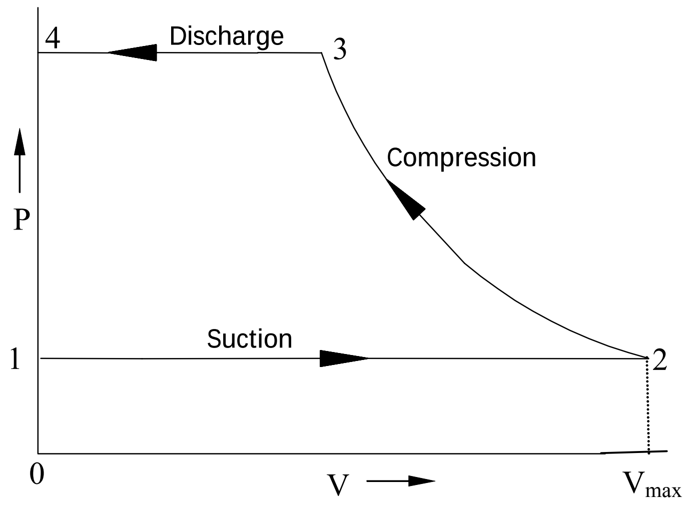



*Figure 1.5: p-v diagram for reciprocating and screw compressor.*

The process 1-2 is the suction process which takes place for one revolution of male rotor. The inducted gas volume of one pair of grooves is the sum of the volumes of male and female rotor grooves. Unlike that in a reciprocating compressor, the suction process in the screw compressor is simultaneously carried out with compression and discharge processes. While a part of a groove is in the suction stage, the rest of the groove undergoes compression. The phenomenon is depicted in Figure 1.7 on a p versus $\theta_m$ plot for better understanding, $\theta_m$ being the angular rotation of the male rotor.

The figure shows that the compression and discharge processes occur simultaneously, the suction process for one revolution of the male rotor starting from the entrapment point '1' and ending in '2'. The compression process occurs between points '2' and '3'. The point '3' is the beginning of discharge process where the pair of compression cavities of the male and female rotors uncovers the discharge port. The point '3' determines the built-in volume ratio on the V-axis and built-in pressure ratio on the P-axis of p-v diagram.

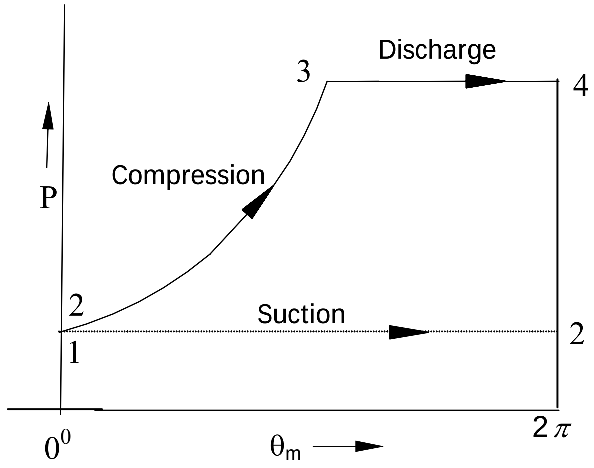



*Figure 1.6 P-$\theta_m$ diagram for twin-screw compressor.*

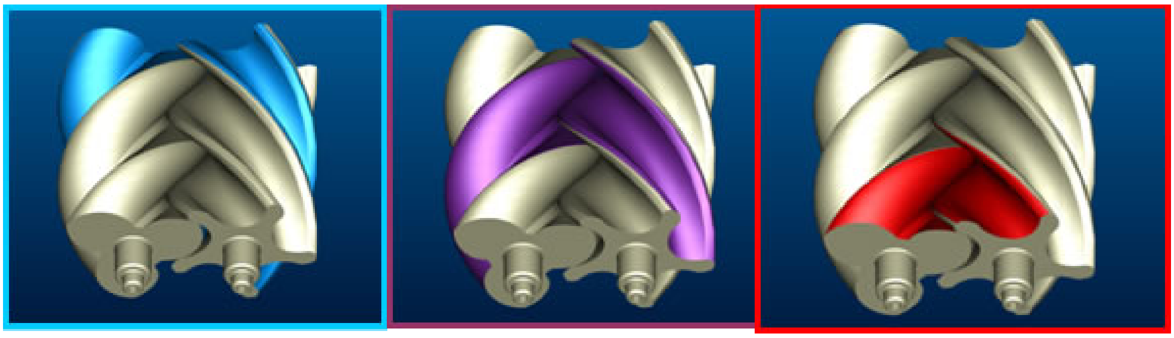


*Figure 1.7 (a)* *Figure 1.7 (b)* *Figure 1.7 (c)*

***Inlet phase*** **:** Figure 1.7 (a) The male and female rotors rotates counter to each other as the lobes of each rotor travel past each inlet port, air is trapped between consecutive lobes and the cylindrical casing. The air move axially forward through outer case and fill the inner lobe space between adjacent lobes.

***Compression phase*****:** Figure 1.7 (b) as rotor mesh air is trapped between rotors and casing. continued rotation progressively reduce space occupied by the air causing compressing air.

***Discharge phase*****:** Figure 1.7(c) compressing continues until the inner lobe space becomes exposed to outlet port through which air discharge through manifold

**Number of lobes and discharge port velocity**

The discharge port velocity generally decreases with increase of number of lobes. This means that the discharge port losses are higher for profiles with less number of lobes. Also, at higher tip speeds, where discharge port losses become significant, the profiles with a lower number of lobes will have poor performance.

**Lobe combination**

Profiles with lobe difference of '1' have marginally better performance than those with differences of '2' at all tip speeds. Based on experimentation and analysis [6] it has been proved that the 5/6 profiles have all the attributes of a good lobe combination in terms of performance, strength, and size. The performance of 3/4 combination profile at high tip speed can be improved considerably by early opening of the discharge port. Thus each design should be optimized according to the individual application.

**Wrap angle**

The decrease in male wrap angle has two effects: the discharge port size increases and the overlap constant decreases. When the wrap angle becomes large, the blowhole expands and the discharge port contracts. Therefore, the internal leakage mass which will re-circulate in the grooves increases and the flow resistance across the discharge port becomes higher. It is evident that the indicated torque increases as the wrap angle becomes smaller, corresponding to the increase in leakage losses. The volumetric efficiency does not vary much as the wrap angle changes. However, the adiabatic efficiency decreases as the wrap angle decreases.

**L/D ratio**

The relative merits of equal and unequal rotor diameters are mostly a function of manufacturing considerations than the design. Only profiles with lobe combinations differing by '2' can be practically made to have equal rotor diameters. This forces the manufacturer to select larger female addendum, which generally results in a large blowhole. A small L/D ratio means larger diameter rotors for a given displacement and larger leakage areas. At higher tip speeds, the leakage areas begin to have less influence and performance for all L/D ratios tend to cluster together. L/D ratios of 1.65 and 1.8 have shown the best overall performance.

**Opening the discharge port early**

One method of improving the performance is to open the discharge port early, i.e. to reduce the built in pressure ratio. This has the effect of increasing the discharge port size at the cost of some back flow compression. However, at higher tip speeds, gas inertia tends to overcome this back flow effect and significant improvement in performance can be achieved. The optimum built in pressure ratio and the other parameters such as discharge port opening angle is function of profile shape, number of lobes, wrap angle, and L/D ratio. Thus, it is necessary to determine the optimum opening angle according to profile shape and operating conditions.

**Symmetric and asymmetric profiles**

There have been extensive rotor profile developments in recent past. Symmetric rotor profiles are predominantly used for oil free screw compressors. The symmetrical circular arc profile in general has large blowhole area. It is always desirable that the rotor profile has smallest blowhole area. Oil injected screw compressor rotors have universally asymmetric rotor profile designs. These types of profiles have shorter sealing lines and smaller blowhole areas.

## 1.5 PERFORMANCE PARAMETERS

Efficient operation of screw compressor is mainly dependent on proper rotor design as discussed above. An important requirement for the successful design of a compressor is the ability to accurately predict the effect of design parameters on performance. Optimum rotor profile, oil injection rate and temperature may significantly differ when compressing different gases or vapours and when working in oil free or oil injected mode of operation.

It is difficult to calculate performance figures from first principles, as the oil is mixed with the gas being compressed. All currently available selection methods are based on empirical data derived from extensive testing on closed loop test rigs. The operation of any compressor requires an input of mechanical work. Most of this mechanical work is eventually converted to heat, principally the heat of compression and also mechanical and aerodynamic friction resulting from the operation.

The power and volumetric efficiencies of a compressor depend on inlet temperature of gas and coolant used. The volumetric efficiency mainly depends on internal leakage and inlet conditions of oil and gas. Although there are no pistons, valves or clearance volume to affect the filling ratio, the leakage losses along the rotors are important. The gas that leaks back to the suction side not only occupies space but also is at higher temperature. It is obvious that the clearances between the rotors and between the rotors and the barrel are important in minimizing the leakage. An increase in clearance of 0.01mm results in decrease of 1% in volumetric efficiency. The dimension of the oil stream also plays an important role on power performance.

The male rotor diameter and its rotational speed determine the tip speed. There is an optimum tip speed to achieve maximum efficiency for each built-in-volume ratio. When the rotor speed is increased keeping the diameter constant, the displaced gas volume per unit time increases and the losses per unit gas volume become relatively smaller. As the losses due to friction and turbulence of the gas increase, the adiabatic efficiency decreases.

The relation between rotor length and diameter has influence on efficiency. Particularly at higher pressure ratios, the discharge port cross section becomes very small and shorter rotor gives less discharge losses. The shape of rotor cross section has considerable influence on the filling ratio. In fixed built-in volume ratio compressors, the compression process always ends at the same point irrespective of the line pressure. This happens when the inter lobe space comes into open contact with the discharge port.

## 1.6 OBJECTIVES OF THE INVESTIGATION

- Generation of different combination of screw compressor profiles in 2D as well as 3D model in GAMBIT and generation of grids to those profiles with minimum skew ness.
- Finding out leakage flow rates in the clearances between male and female, female and casing, male and casing using moving reference frame in FLUENT.
- Simulation of screw compressor.

## 1.7 ORGANIZATION OF THE THESIS

The thesis has been arranged into six chapters. Chapter 1 (this chapter) deals with a general classification of compressors and introduction to the twin screw compressor in more detail and enumerates the objective of the present investigation. In chapter 2, a brief review of relevant literature covering theoretical and experimental studies has been presented. Chapter 3 covers the CFD package material so that reader can understand the model preparation this chapter mostly two parts out of these first part describes the GAMBIT facilities for modelling and grid preparation where as second part is the details of equation solving steps and control the solving process.

Chapter 4 covers generation of different combinations of screw compressor profiles in 2D and 3D, generation of mesh for different profiles in GAMBIT. Moving boundary conditions are used to calculate tangential velocities around the rotors. Leakage flow rates of screw compressor are found in FLUENT. Results and Discussions are mentioned chapter 5. The final chapter is confined to some concluding remarks and for outlining the scope of future work.

# CHAPTER 2

## LITERATURE REVIEW

Compressor Design

Flow and Leakage

CFD on screw compressor

# LITERATURE REVIEW

Around 1928 gear theories applicable to gas compression were developed by Nahuse of Tohoku University, Japan, and by the Russian engineer Novikov. These theories, however, were not used practically in compressor technology at that time. The first practical compressor was invented by Lysholm in 1934 and was mainly developed by SRM (Svenska Rotor Maskiner AB) of Sweden. Lysholm's compressor, which had a 3/3 profile combination, was produced in 1934. By 1937, a 4/6 Profile combination machine had been developed, and in 1938, such a compressor was manufactured in collaboration with Lysholm by James Howden & company of Glasgow, Scotland.

Since the late 1950's, it has received practical applications for industrial use. The Scottish engineer Duncan Laing at James Howden & company tested the first operating screw machine in 1955.

In the 1960's, the twin-screw compressor come to existence, providing high capacity with reduced size and cost, together with an option to operate with high compression ratios allowing single stage systems for gas compression and low temperature refrigeration requirements.

More recently, there has been a lot of research activity at the City University of London (at www.city.ac.uk) on methods to improve rotor design as well as volumetric and adiabatic efficiencies. Despite the rapid growth in screw compressor usage, public knowledge of the scientific basis of their design is still limited. In this chapter, relevant published literature is reviewed on the subject, focussing on design and performance.

Mathematical modelling, experimental validation, design of critical components, complete screw machine design, product development, training in machine design, advanced computerized design tools, machine process modelling, 2-D and 3-D computational fluid dynamics, modern experimental techniques, computerized data acquisition, rotor and compressor optimization are the essential stages needed for appropriate screw compressor development [7-9]. Screw compressor rotors of various profiles can be conventionally manufactured today with small clearances at an economic cost and the internal leakages have been reduced to a small fraction of their earlier designs

Screw machines have been used today for different applications both as compressors and as expanders [8]. They operate on a variety of working fluids. The working fluid may be gas, dry vapour, or multi phase mixture with phase change taking place within the machine. These machines may operate oil flooded or with other fluids injected during compression process. In the field of air and gas compression, screw machines are continuously replacing reciprocating and vane compressors and a dramatic increase in its applications in the field of refrigeration compressors are expected in the next few years.

Comparative investigations between rotary and reciprocating compressors have been presented by Kaiser and by Villadsen. In their experimental investigations, various reciprocating and rotary compressors of comparable capacities have been analyzed on the basis of their thermodynamic and mechanical losses. The authors have explained the differences and have concluded that that both types of positive displacement compressors have their own merits, and that they complement each other to the extent that they may often be combined in one plant to obtain the most energy efficient installation under variable operating conditions.

Different operational modes of twin-screw compressors have been explained by Sjoholm, who concluded that screw compressors could be adapted to every specific need without losing its favourable characteristics as a heavy-duty machine with high performance.

## 2.1 COMPRESSOR DESIGN

Screw compressor designs have gradually evolved through history and the trend has been to realise as small machine as possible to meet the required performance. This means that rotor tip speeds are as high as possible with the limits imposed by efficiency requirements. Wherever possible, rolling element bearings should be used to permit the small clearances necessary instead of journal bearings.

Similarly the ports are to be made as large as possible to minimize suction and discharge gas speeds and consequent pressure losses. The oil injection port position on the compressor casing is to set at the point where thermodynamic calculations show the gas and oil inlet temperatures to coincide.

To minimize flow losses in the suction and discharge ports, the suction port should be positioned in the housing so as to let the gas enter with the fewest possible bends and the gas approach velocity kept low by making the flow area as large as possible. The discharge port size is first determined by estimating the built in volume ratio required for optimum thermodynamic performance.

The casing should be carefully dimensioned to minimize its weight, containing reinforcing bars across the suction port to improve its rigidity at higher pressures. Bennewitz suggests a comprehensive method for design, manufacture and quality control using computer assisted manufacturing.

## Geometrical Parameters

Screw compressor rotor geometry plays a crucial role in its design and performance. For a given application, there are a number of design possibilities. But, normally only a few designs can adequately fulfil the basic requirements of reliability, high performance and low cost. The rotor profile not only affects the performance and torque distribution between the rotors, but also the axial and radial loads. The important geometrical parameters of a compressor and their influence on performance have been studied by Tang, Sjoholm and Singh [10-12].

Zhang and Hamilton studied the effect of main geometric characteristics such as compression volume curve, sealing line length, flute area, wrap angle and blowhole area. Mathematical models of these parameters were formulated to develop the manufacturing software. Tang and Fleming [10] studied the effect of relative blowhole area and relative contact line length on performance and suggested some methods for geometrical parameter optimization.

Singh and Bowman [6] examined the effect of some of the geometrical parameters like number of lobes, wrap angle, L/D ratio and opening of discharge port early for a particular profile shape. The authors focussed more attention on fundamental aspects of the design process. Number of designs was created and their dimensions were normalized to male rotor diameter of the 5/6 profile to get identical displacement per unit revolution in all types of rotors. The authors applied generalized mathematical modelling to calculate the geometrical characteristics and presented the results. It has been observed from the results that the contact or interlobe sealing line length increases strongly with the number of lobes, which has an adverse effect on performance at low male rotor tip speeds and high pressure ratio.

A parametric study on twin screw refrigeration compressor performance has been carried out numerically by You *et al* [13] for optimum rotor geometry with four commonly used lobe combinations and five different lengths to diameter ratios and wrap angles. The four lobe combinations considered in the study are 4/5, 4/6, 5/6, 5/7; the length to diameter ratio ranges from 1.0 to 2.2, and the male wrap angle ranges are from $250^0$ to $300^0$. Apart from performance parameters, factors like rotor deflection, bearing load and its life and inter rotor contact forces are also included. The influence of geometrical parameters has been discussed and suggestions on choosing optimum parameter combinations presented. Female rotor deflection has been calculated since it is several times higher than that of the male rotor. It has been found that the 4/5 combination is an excellent choice since it has the smallest size and lowest weight, while the 5/6 design has the lowest input torque. The results, however, show that these two combinations have relatively larger deflections than the 4/6 and 5/7 combinations.

The authors have concluded that 4/6 and 5/7 profile combinations are the better choices for high pressure applications. Also, the 5/6 combination showed relatively higher isentropic indicated efficiency for L/D ratios up to 1.7. The 5/7 combination showed that the performance of this combination is very close to that of the 5/6 combination for L/D ratio above 1.7, and that their deflections are much smaller as shown in Figure 2.1. It has been concluded that the 5/6 combination is more appropriate for L/D ratio above 1.7, particularly for high pressure ratio applications.

**Rotor Profile Generation**

Any design process can be made more reliable with mathematical modelling and numerical simulation. With the advancement of computational facilities, prototyping has been reduced to a minimum. The design of screw compressor is interactive and the measured performance of the compressor has to be compared with that specified in advance. Usually this is achieved by testing a prototype system and modifying the design until it yields satisfactory results.

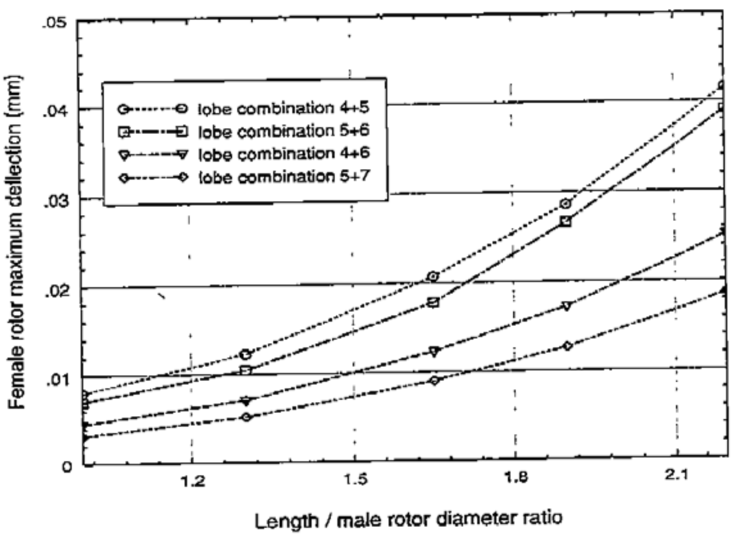



*Figure 2.1: Variation of female rotor deflection with L/$D_m$ ratio [13]*

An efficient screw compressor needs a rotor profile which has a large flow cross-section area, short sealing line length and a small blowhole area, to achieve higher flow rate for the same rotor size and speed. Short sealing lines and small blowhole reduce leakage. Higher flow and smaller leakage rates will increase volumetric efficiency. This, in turn, increases the adiabatic efficiency, because less power is wasted in the compression of leaked gas, which is re-circulated internally. As precise manufacturing permits the rotor clearances to be reduced despite oil flooding, the likelihood of direct rotor contact increase on reduced clearance. Hard rotor contact leads to deformation of the female rotor due to increased contact forces and ultimately rotor seizure. Hence, the profile should be designed so that the risk of rotor seizure is eliminated.

Relatively few publications are available on screw compressor design since their large scale manufacture began only in the early nineteen seventies as a result of the introduction of the 'A' Profile by the Swedish company SRM. Arbon [1] dedicated his book exclusively to twin shaft compressors and their applications with limited details. Xing published a comprehensive book on this topic but it is written in Chinese language and is generally not available outside China. Only recently, Stosic *et al* [9] have published a book emphasizing mathematical modelling and performance.

Stosic and Hanjalic [14-17] presented a general algorithm for generation of screw rotor profile and related machine geometry. The method is convenient for the design of screw rotors as well as for improvement of existing rotors. A rack based procedure, capable of generating modern screw rotor profiles has also been included. The main advantage of the algorithm lies in its simplicity, and its capacity to enable ordinary mechanical engineers to create a variety of profiles, a privilege which was previously shared only by a limited number of exclusive specialists. The conjunctive condition has been solved numerically, thereby introducing a variety of primary arc curves. The approach has simplified the design procedure since only primary arcs need to be given, the secondary arcs being automatically generated.

Singh and Onuschak [18] developed a rapid, flexible and comprehensive computer assisted technique to analyze twin-screw rotor profile generation methods. The strength of this method lies in the reduction in time taken by the whole process from profile generation to performance prediction. This powerful profile generation tool can be used in many ways. It has been instrumental in inventing and analyzing entirely new profiles and modifying existing profiles to match particular applications, optimizing geometrical parameters of machines and understanding the importance of different leakage areas and power loss. It has also been used for generating input data such as contact line length and blowhole area for performance prediction programmes. The flow chart of program sequence used for profile generation is shown in Figure 2.2. The model has been used several times successfully for all kinds of applications including analysis and evaluation. The profile types investigated had a wide range of shapes, number of male/female lobes (3/4 to 6/8 and in between), wrap angles (150 to 350$^0$), $L/D_m$ ratios (0.8 to 2) and built in pressure ratios (4 to 10).

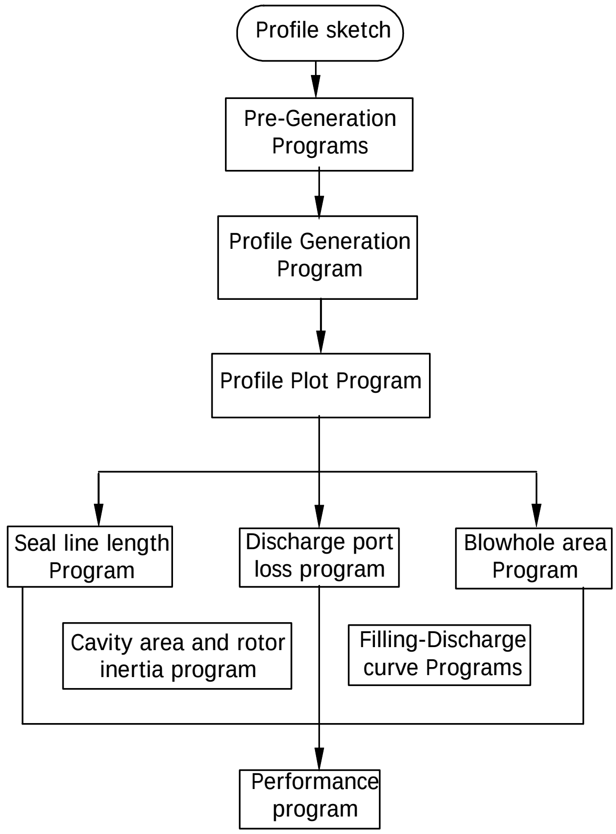



*Figure 2.2: Flow chart of computer program to generate screw profile [24]*

New designs of rotor profiles have been introduced considering modern design concepts such as larger opening of the suction port and early exposure of the discharge port together with improved bearing systems and seal specifications to maximize work efficiency.

Prototype compressors have been tested by the authors and compared with the best compressors currently available in the market. The maximum measured specific power input at a delivery pressure of 8 bar was 5.6 kW /($m^3$/min), which was less than the published value for any other equivalent compressor manufactured at that time.

McCreath *et al* [19] have published a paper that describes the development of two highly efficient oil free screw compressors designed for dry air delivery. Their design is based on the use of rack generated 3/5 rotor profiles. The optimum rotor size and speed, together with the shape and position of the suction and discharge ports, were determined by mathematical modelling. The model took full account of the limitations imposed by selection of bearings and seals required to maximize endurance and reliability. Xing *et al* developed a software package to design twin-screw compressors. The package was used to calculate the rotor profile, geometrical characteristics, thermodynamic performance, and forces on rotor teeth, rotor shape and cutter shape. A user friendly interface and some powerful post processing programs were also included in the package. The same package has been used for improving the performance of an existing machine.

Rotors of helical screw compressors present a challenging manufacturing problem. This is due to complex profiles and the fine tolerances necessary for machining the minimal running clearances required for efficient operation. The rotors may vary in length to diameter ratio, male to female diameter ratio, male to female lobe ratio, diameter, profile, and host of other parameters.

Mould *et al* presented a method which proceeded directly from an analytical description of the profile to a computer generated cutter template using a numerically controlled contouring grinder having an accuracy and repeatability in the region of 1 μm. They present a method for describing the tool paths in CNC machining starting with the mathematical theory of the design of milling cutters.

Zhou developed a computer aided design method for profile generation, meshed line and contact line plotting, generation of pressure distribution diagram, blowhole area calculation, and milling cutter profile calculation and plotting.

Xing introduced a new theoretical approach and practical application of a CAD system to machining of twin-screw compressor rotors. It comprises of procedures for calculation of geometrical parameters of rotors, simulation of the compressor working process and optimization of the design parameters. The design process has been used to determine the manufacturing and operating parameters for several air and refrigeration compressors.

Computerized design, profile generation and simulation of meshing of rotors have been done by Faydor and Feng . They have covered the conjugation of surfaces, investigation of influence of misalignment on the backlash between the surfaces, synthesis of rotor surfaces with two lines of contact and avoidance of singularities.

Fluid leakage across the contact line between two conjugate helical surfaces is the major concern in terms of machine efficiency. To characterize the geometrical shape of the leakage path and to find the contact line, an average comprehensive radius of curvature (ACRC) analysis has been proposed by Xiao *et al* .

A numerical approach is accordingly developed which is suitable for microcomputer implementation and used as a part of the CAD and performance analysis package for screw compressors.

**Design Optimization**

Continuous increase in demand for efficient screw compressors requires that the designs are tailor-made to address to varying duty, capacity and manufacturing capability. A suitable procedure for optimization of screw compressor shape, dimension and operating parameters needs to be developed which will lead to the most appropriate design for a given compressor duty. An optimization technique has been developed and applied to the design of refrigeration twin-screw compressors. The authors also measured the operating parameters as well as rotor and compressor parameters such as wrap angle, L/D ratios and slide valve specifications.

You *et al*, gave special attention to the male rotor crest range angle and female rotor addendum, since these parameters have a greater influence than all other rotor tip parameters on optimum lobe tip design.

Xing *et al* , carried out research on the design of a new generation of refrigeration compressors. The rotor profile and other design parameters were optimized with the help of the software package SCCAD.

Stosic *et al* [15, 20,21] carried out the design of a family of efficient oil flooded twin screw air compressors using a software package which included almost every aspect of rotor profiling and compressor thermodynamic and geometric modelling with the capacity to transmit calculated output directly into a CAD drawing system. They have also designed a family of highly efficient screw rotors based on rack-generated profiles as shown in Figure 2.3, which can be used to replace standard asymmetric profiles.

Optimization of a single stage compressor for oil-free and oil-flooded air compression and refrigeration applications has been developed by the authors as shown in Figure 2.4. Therefore, extensive calculations have been carried out by Stosic *et al* [21] before a final decision on the compressor design has been made.

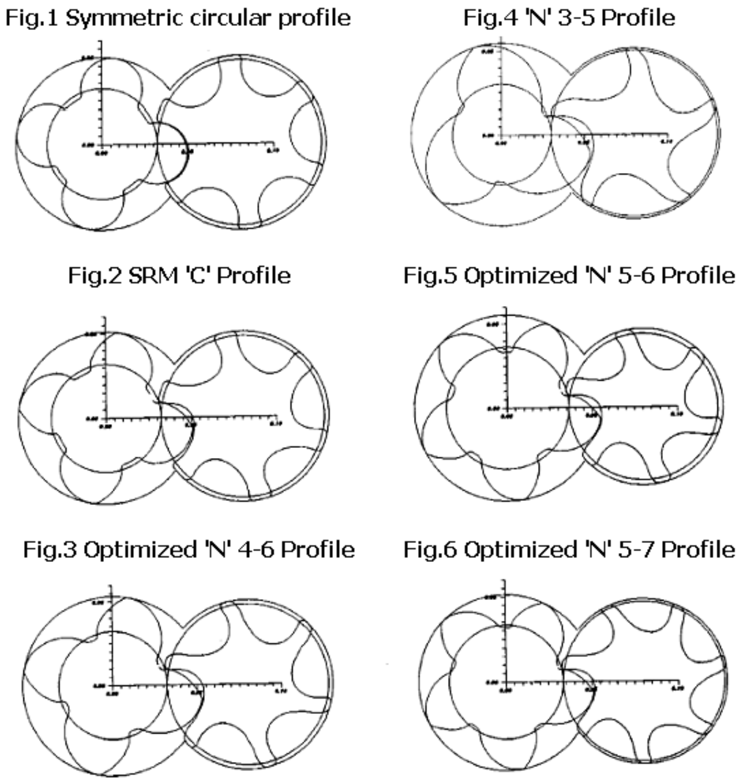



*Figure 2.3: Optimized Screw Rotor Profile Designs [22]*

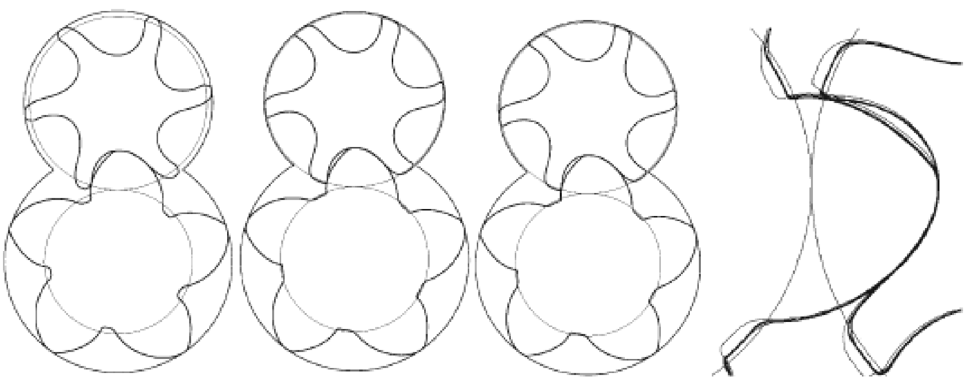


*Figure 2.4:Rotor profiles optimized for oil-free and oil-flooded air and refrigeration compressor duty [21]*

## 2.2 FLOW AND LEAKAGE

To minimize the flow losses in the suction port, the suction port should be positioned on the housing to let the gas enter with the fewest possible bends and the gas approach velocity should be kept low by making the flow area as large as possible. The discharge port size is determined by estimating the built in volume ratio required for optimum thermodynamic performance. Most of the theoretical work done on gas leakage in compressors is based on quasi one dimensional steady state models.

The shape and position of the suction and discharge ports influence the dynamic losses. The position of oil injection port and the quantity of oil injected into the working chamber affect both the outlet temperature and the power consumption. Dynamic flow loss in the suction chamber is a significant factor in reducing screw compressor efficiency but can only be roughly estimated during compressor design step due to the simplified methods of analysis used to account for these effects.

Today, computational fluid mechanics based analysis is used to estimate velocity, pressure, temperature and concentration fields within a compressor far more accurately than was done earlier. Stosic *et al* applied this technique to design a suction port with minimized flow losses. The function of the discharge port on a screw compressor is to provide an exit of the gas and oil from the flutes of the rotors. The volume of flutes occupied by the gas at the beginning of discharge relative to the total volume of the cavities gives the volume ratio of the compressor. An error in calculation of these volumes will lead to error in calculation of volumetric and adiabatic efficiencies. Sjoholm and Muralidhar have published a paper [22] on measurements on the axial discharge port and its geometric tolerance and have suggested a method to minimize discharge port losses.

Leakages are generally considered to be one of the major sources of efficiency loss in screw machines. Rotor clearances are mainly responsible for leakage. Clearances must be minimized to obtain high volumetric and power efficiencies. The gas that leaks back to the suction side not only occupies space but is also at higher temperature. It is obvious that manufacturing tolerances on rotors and the barrel are very important. An increase in tolerance by 0.01mm results in an increase of 1% in volumetric losses . The dimension of the oil stream also plays an important role.

The actual profile of the compressor rotors and their engaged clearance have a great effect on volumetric efficiency and noise. Dynamic measurements on compressor rotor pair in connection with the working process and analysis of rotor profile including engaging clearance has been described in detail by Xiong [23]. Hangiqi and Guangxi developed a computer model considering the effect of discharge port position on compressor performance at various working conditions.

For practical computation of the effects of leakage during the compression process, it is convenient to distinguish between two types of leakages according to their direction in the working chamber: gain and loss leakages. The gain leakages flow into a compression cavity from the discharge plenum and from the neighbouring working chambers which are at higher pressures. The loss leakages leave the chambers towards the suction plenum and to the neighbouring chambers having lower pressures.

The leakage of gas and oil mixture takes place through interlobe clearance, blow holes and gaps between the plate and the rotors at the discharge end. At the lobe tip, the clearance fills with oil due to the action of centrifugal forces, and only oil leakage takes place which can be calculated by using the equation of incompressible viscous flow through a narrow gap. The amount of oil leakage depends on the gap between the rotor tip and the housing, and the sealing line length. The sealing line length depends on the rotor turning angle. As the rotor starts rotating, the sealing line length decreases. It has the maximum value at the beginning of compression and is zero when the discharge process is completed.

Presence of leakage triangles (blow holes) is an inevitable consequence of the rotor profile geometry. During compression, two blowholes called leading blowhole and lagging blowhole are formed. Both are of same cross sectional area. Leakage of oil-gas mixture into the working chambers is from the leading blowhole and out of the working chamber is through lagging blowhole.The fluid leakage across the contact line of conjugate helical surfaces is the major concern in terms of machine efficiency. An average comprehensive radius of curvature (ACRC) analysis was proposed by Xiao and Liu to characterize the geometrical shape of leakage path. A numerical approach was developed for the analysis. It is suitable for microcomputer implementation and is used as a part of the CAD system and performance analysis package.

Leakages are calculated basing on the assumption that gas and oil are uniformly mixed and thermally isolated from the surroundings. The oil-gas mixture leakage rate is calculated using a standard formula for compressible fluids flowing through a convergent nozzle. The flow and blockage coefficients identify the effects of refrigerant viscosity and the sealing function of the lubricating oil used.

The properties of gas and oil mixture coming out from the leakage paths and through the discharge port need to be known a priori. But in the absence of accurate thermodynamic data, they are usually determined by comparison of theoretical models and experimental data. By comparison with laboratory tests, Sangfors suggested the following assumptions for different types of leakage paths: (i) the gas/oil mixture is homogeneous in all leakage paths and (ii) the gas/oil mixture ratio is same in all leakage paths except at the lobe tip clearance, and is equal to the mixture ratio in the discharge port.

The leakage rates were calculated basing on the above assumptions. Due to the presence of oil, exact determination of specific heat ratio is rather difficult. Fujiwara and Osada defined an apparent ratio of specific heats and a modified gas constant and used them in their simulation models.

The average leakage area is determined by multiplying the sealing line length with an average gap (clearance) for each type of leakage [24]. The average gap/clearance is determined from actual clearance measurements on the compressor. The discharge or flow coefficients are empirically selected for each leakage to account for the presence of oil. The flow of oil gas mixture through the leakage paths is in two-phase. Exact determination of physical properties of oil-gas mixture is difficult. Based on extensive test data the oil-gas mixture properties have been determined by Sangfors.

Vimmer suggested the essential steps for numerical simulation of compressible inviscid flow in a sealing gap starting from the description of a mathematical model to its final numerical solution. For solution of the system, the cell centred finite volume formulation of the explicit two-step MacCormac scheme with Jameson’s artificial dissipation was used. Zaytsev and Ferreira presented a one dimensional leakage flow model for two phase ammonia-water twin-screw compressor. The governing equations were solved using a finite difference method. Results of the solution were used for calculation of shear stress and friction between the rotors and the housing. Comparison of the proposed leakage flow model with the results of the isentropic converging nozzle model showed that the latter predicts up to two times higher leakage mass flow rate.

Leakage losses are directly proportional to the effective leakage areas. A method to determine the aggregate leakage through each path individually over a complete compression cycle is required to compute the gas leakage loss. Fleming and Tang constructed a mathematical model for thermo-fluid processes suitable for leakage calculation. Analytical techniques were proposed for different rotational speeds and experimental methods were suggested for optimized compressor design.

The design of the rotor-casing assembly is primarily controlled by the consideration of internal leakage. Optimised rotor geometry and proper choice of clearances reduce leakage losses. To study the influence of rotor clearances on compressor efficiency, computational fluid dynamics (CFD) analysis has been carried out using FLUENT package [25]. The analysis has been done for static rotors at different positions.

The results obtained have shown that the size of the clearances and the geometry of rotor lobes have significant effect on gas leakage and distribution of leakage over the three main leakage paths.

Leakage experiments on a running twin-screw compressor were carried out by Prins *et al*. In their experiment, the indicated diagram was measured for different sizes of rotors and sealing line gap. Lee *at al* have numerically analysed leakage performance of a screw compressor, assuming turbulent flow through a plain seal with oil injection.

The effects of parameters such as rotation speed, injection speed, clearance ratio, injection angle and axial injection location on flow pattern and leakage performance have been investigated. The authors concluded that with oil injection, the total leakage of compressed gas can be reduced to acceptable levels.

## 2.3 CFD ON SCREW COMPRESSOR:

The finite volume method is a powerful CFD numerical technique, which allows fast and accurate solution of the governing differential equations for fluid flow within complex geometries. However, an acceptable grid system, which describes shapes accurately, must be available in order for the method to be used. Structural grids are generated by Ahmed Kovacevic, Boundary Adaptation in Grid Generation for CFD Analysis of Screw Compressors [26]. the use of 3-D numerical modeling of the compressor fluid flow and structure deformation, these factors can be predicted more precisely and hence losses within the machine can be minimized at the design Stage[27].The numerical analysis is conducted using Computational Fluid Dynamics (CFD) software commercially available, FLUENT [28].

# CHAPTER 3

## OVERVIEW OF FLUENT CFD PACKAGE

GAMBIT

Numerical solving technique

Modeling Flows in Moving and Deforming Zones

Problem solving steps

# OVERVIEW OF FLUENT CFD PACKAGE

The availability of affordable high performance computing hardware and the introduction of user-friendly interfaces have lead to the development of commercial CFD packages. Several general-purpose CFD packages have been published in past decade. Prominent among them are: PHONICS [21], FLUENT [12], SRAT-CD [19], CFX [20], FLOW -3D and COMPACT. Most of them are based on the finite volume method.

Among these as mentioned FLUENT is very leading engineering software provides a state of the art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving the flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/ quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge, and mixed (hybrid) meshes.

Fluent also allows refining or coarsening the required mesh based on the flow solution. FLUENT also allows refining or coarsening the required mesh based on the flow solution. FLUENT consists of two main parts. First part is called GAMBIT and second part is called FLUENT the solver.

One can generate the required geometry and grid using GAMBIT. Also one can use T grid to generate a triangular, tetrahedral or hybrid volume mesh from the existing boundary mesh (created by GAMBIT of a third party CAD/CAE package).

Once a grid has been read into FLUENT, all refining operations are performed within the solver. These include the setting boundary conditions, defining fluid properties, executing the solution, refining the grid viewing and post processing the results.

## 3.1 GAMBIT

Take care to insure that you are in the correct directory. Fire up gambit from the command prompt by typing gambit *filename.* The first thing that you should do is to specify which solver you need from the Solver menu. Choose 'Fluent 5/6'. This will determine what type of menu popup throughout your session.

**Generate a grid**

There are two ways of generating a mesh. Gambit calls them 'top down' or 'bottom-up' in the user manuals. These instructions are bottom-up. You will create vertices upon which the edges will be built upon. Connecting edges will create a face. Connecting faces will create a volume (3D). Once the face or volume is created, a mesh can be generated on it. For this example, we will stick to 2D, node -> edge ->face-> mesh. Remember to save and save often.

**Vertex:**

There are four buttons under the word OPERATIONS in the top right corner of Gambit. They are, from left to right, the *geometry, mesh, zones* and *tools* command. At this time, click on the geometry button. Note: most of the buttons in Gambit toggle off and on. The blank space under the buttons on the right hand side is now showing more buttons and windows. Directly under OPERATIONS is GEOMETRY with 5 buttons: *vertex, edge, face, volume,* and *groups.* Click on the *vertex* button.

By this time, you will have noticed that as you move the mouse over the function buttons a window near the bottom of Gambit tells you what that button does. Use this function to familiarize yourself with the various buttons in Gambit.

Once you have clicked on the *vertex* button more buttons appear below. Click on the button directly below the vertex button called Create Vertex. A floating window called Create Real Vertex appears below. Here you may enter the coordinates of the vertices in your problem. Don't worry about local coordinates at this time. Enter your coordinates in the global area. As you enter in the vertices, they will show up as white X’s in the view area. If you cannot see them they may be outside of your viewing area. To remedy this, click on the Fit to Window button, the top left big button in the GRAPHICS/WINDOWS CONTROL area (near bottom right).

If at any time you wish to undo the command you just did, look for the button that has the arrow that is 'spinning' from right to left. The Undo command can undo more than one command, just keep clicking.

For more complicated geometry, such as an airfoil, the vertex data can be imported. Go to File -> Import -> Vertex Data. Enter the path to the file or us the browser. The data file

Gambit can read has to have the file extension .dat. The format of the data in the file must be tab or space delimited.

Most of the data downloaded from the internet will typically need to be modified. There should be no text in addition to the data and a column of zeros for the z -axis will need to be added.

**Edges:**

Once the vertices are created, you want to create edges connecting them. Under GEOMETRY, click on the edge button (second from left). When the EDGE buttons pop up, right click on the first button on the left. A drop down list will appear giving different options for the edge type. When one of these options is selected a floating window will be displayed. To create smooth curved edges use the NURBS option. There are two methods for the NURBS, interpolate and approximate.

The approximate method with a tolerance of zero will give a smooth curve. To select the vertices for the NURBS line left click the up arrow on the right side of the yellow vertices box. Select the vertices with the mouse and click on the ---> button. Once the vertices are selected, the final one will turn red and the others will turn pink. If the vertices are the ones you want to connect with an edge then click Apply in the floating window. An edge will appear in yellow. Use this procedure to create an edge for the top and bottom of the airfoil and the control volume.

**Face:**

Under GEOMETRY, click on the *face* button (third from left). When the FACE buttons pop up, click on the first button on the left: Create Face. A floating window called Create Face From Wireframe will appear. Selecting an edge is the same as selecting a vertex. Hold the shift key down and left click on the edge. The edge will turn red. Select a second edge: the first will turn pink and the second will turn red. Select all edges comprising the face and click Apply in the window. A face will be created; its color is light blue. To create a single face from two faces use the Boolean Operations Subtract option.

**Mesh:**

A mesh can now be created on the face. Under the OPERATION button, click on Mesh Command button. Where the word GEOMETRY used to be, the word MESH will appear with five buttons: *boundary-layer, edge, face, volume* and *group*. You want to mesh the face that you have just created, so click on *face*. Click on the top left button in the FACE menu area, the button is called: Mesh Faces. This will cause the Mesh Faces floating window to pop up. Let everything stay at its default, select the face and click Apply. Gambit may hesitate while it's thinking and then you will see the mesh in yellow. You can play around with mesh spacing but keep the elements and type at Gambits default setting.

**Boundary Conditions:**

You can set or change the boundary conditions in Fluent but you can also do it in Gambit, in fact, it's a little bit easier. Up in the OPERATIONS menu; click on the Zones button. Under the word ZONES two buttons will appear: Specify Boundary Types and Specify Continuum. Click on the Specify Boundary Types button. A floating window called Specify Boundary Types will appear. Make sure that at the top of this window the solver name 'Fluent 5/6' appears, if not go to the solver menu and choose 'Fluent 5/6'. You must have this correct as different solvers specify BC's differently.

Change the Entity pop down menu to edges. Select the edge that will be the velocity inlet and under the Type pop down menu choose Velocity Inlet. It is recommended that you label the different edges. This will help you keep track of them in the Fluent output reports. The labels must be one word, i.e. no spaces or tabs. To finish creating the BC click *Apply*. Now select the edge that will be the outlet and choose Outflow. The top and bottom edges of the airfoil and control volume are Walls. There is a list at the top of this window that should reflect the two BC's that you have created.

**Save and Export:**

The file that you have been saving to throughout the session is a Gambit file and is different from a mesh file. To create the mesh file for Fluent to import click on File-> Export-> Mesh. The next pop up window will have file type (UNS/RAMPANT

/FLUENT 5) and file name. Type in the name as you please but keep the .msh file name extension. If the geometry is 2D, then check the box “Export 2d Mesh”.

## 3.2 NUMERICAL SOLVING TECHNIQUE

FLUENT in general solve the governing integral equations for the conservation of mass and momentum, and (when appropriate) for energy and other scalars such as turbulence and chemical species. Usually control volume based technique is used that consists of:

- Division of the domain into discrete control volumes using a computational grid.
- Integration of the governing equations on the individual control volumes to construct algebraic equations for the discrete dependent variables (“unknowns”) such as velocities, pressure, temperature and conserved scalars.

Linearization of the discretized equations and solution of the resultant linear equation system to yield updated values of the dependent variables.

**Solution Methodology**

FLUENT allows choosing either of two numerical methods:

- Segregated solver
- Coupled solver

The two numerical methods employ a similar discretization process (finite volume), but the approach used to linearize and solve the discretized equation is different**.**

**Segregated Method**

Using this approach, the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps illustrated below:

1. Fluid properties are updated, based on the current solution. (if the calculation has just begun, the fluid properties will be updated based on the initialized solution).
2. The $u, v$ *and* $w$ momentum equations are each solved in turn using current values for pressure and face mass fluxes, in order to update the velocity field.

3. Since the velocities obtained in step 2 may not satisfy the continuity equation locally, a “Poisson-type” equation for the pressure correction is derived from the continuity equation is then solved to obtain the necessary corrections to the pressure and velocity fields and the face mass fluxes such that continuity is
4. Where appropriate, equations for scalars such as turbulence, energy, species and radiation are solved using the previously updated values of the other variables.
5. When inter-phase coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation.

These steps are continued until the convergence criteria are met.

**Coupled Method**

The coupled solver solves the governing equations of continuity, momentum and (where appropriate) energy and species transport simultaneously (i.e., coupled together). Governing equations for additional scalar will be solved sequentially (i.e., segregated from one another and from the coupled set) using the procedure described for the segregated solver. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps outlined below:

1. Fluid properties are updated, based on the current solution. (if the calculation has just begun, the fluid properties will be updated based on the initialized solution).
2. The continuity, momentum and (where appropriate) energy and species equations are solved simultaneously.
3. Where appropriate, equations for scalars such as turbulence and radiation are solved using the previously updated values of the other variables.
4. When interphase coupling is to be included, the source terms in the appropriate continuous phase equations may be updated with a discrete phase trajectory calculation.
5. A check for convergence of the equation set is made.

These steps are continued until the convergence criteria are met.

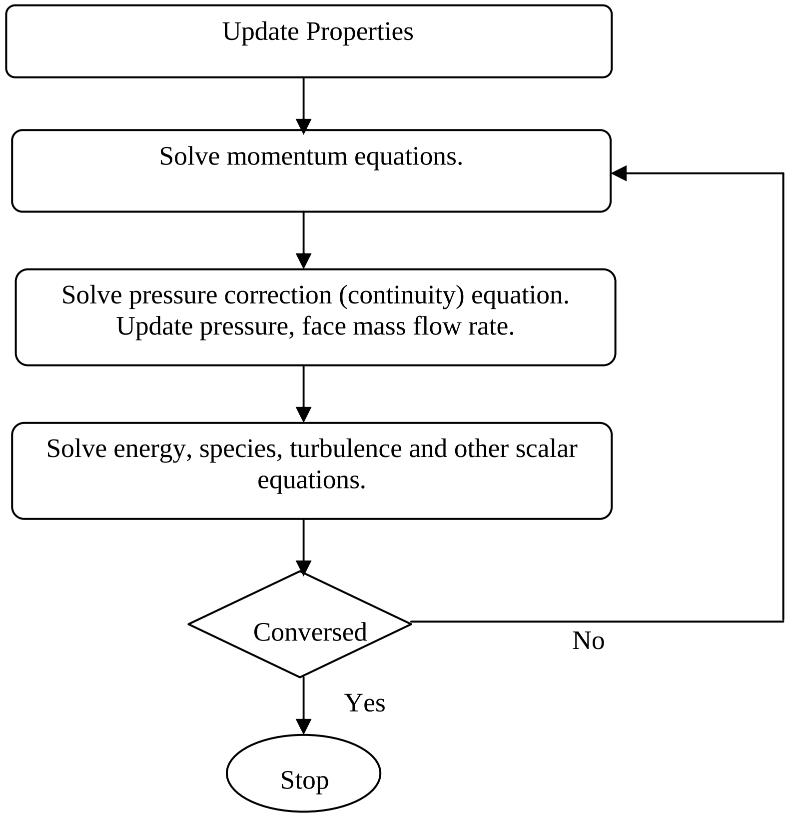



*Figure 3.1: Overview of the Segregated Solution Method*

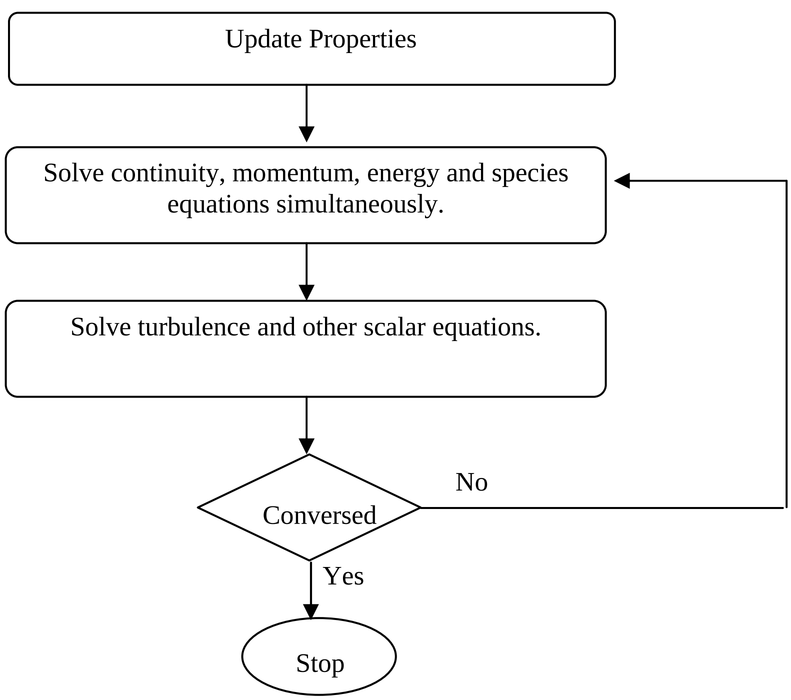



*Figure 3.2: Overview of the Coupled Solution Method*

**Linearization: Implicit & Explicit**

In both the segregated and coupled solution methods the discrete, non-linear governing equations are linearized to produce a system of equations for the dependent variables in every computational cell. The resultant linear system is then solved to yield an updated flow-field solution.

The manner in which the governing equations are linearized may take an “implicit” or “explicit” form with respect to the dependent variables (or set of variables) of interest. By implicit or explicit we mean the following:

- **Implicit:** for a given variable, the unknown value in each cell is computed using a relation that includes both existing and unknown values from neighbouring cells. Therefore each unknown will appear in more than one equation in the system, and these equations must be solved simultaneously to give the unknown quantities.
- **Explicit:** for a given variable, the unknown value in each cell is computed using a relation that includes only existing values. Therefore each unknown will appear in only one equation in the system, and the equations for the unknown value in each cell can be solved one at a time to give the unknown quantities.

In the segregated solution method each discrete governing equation is linearized only by implicitly with respect to that equations dependent variable. This will result in a system of linear equations with one equation for each cell in the domain. For example, the x-momentum equation is linearized to produce a system of equations in which u velocity is the unknown. Simultaneous solution of this equation system (using the scalar AMG solver) yields an updated u velocity field.

In the coupled solution method user have a choice of using either an implicit or explicit Linearization of the governing equations. Governing equations for additional scalars that are solved segregated from the coupled set, such as for turbulence, radiation etc., linearized and solved implicitly using the same procedures as in the segregated solution method.

If one choose the implicit option of the coupled solver, each equation in the coupled set of governing equations is linearized implicitly with respect to all dependent variables in the set. This will result in a system of linear equations with N equations for each cell in the domain, where N is the number of coupled equations in the set. For example, Linearization of the coupled continuity, *x, y, z* momentum and energy equation set will produce a system of equations in which $\rho, u, v, w$ *and* $T$ are the unknowns. Simultaneous solution of this equation system (using the block AMG solver) yields at once updated pressure, *u, v, w* velocity and temperature fields.

If one chooses the explicit option of the coupled solver, each equation in the coupled set of governing equations is linearized explicitly. As in the implicit option, this will result in a system of equations with N equations for each cell in the domain. And likewise, all dependent variables in the set will be updated at once. However, this system of equations is explicit in the unknown dependent variables. For example, the x-momentum equation is written such that the updated x velocity is a function of existing values of the field variables. Because of this, a linear equation solver is not needed. Instead, the solution is updated using a multi stage (Runge-kutta) solver.

In summary, the coupled explicit approach solves for all variables ($\rho, u, v, w, T$) one cell at a time.

## 3.3 MODELING FLOWS IN MOVING AND DEFORMING ZONES

FLUENT can model flow involving moving reference frames and moving cell zones, using several different approaches, and flow in moving and deforming domains (dynamic meshes). Solving flows in moving reference frames requires the use of moving cell zones. This cell zone motion is interpreted as the motion of a reference frame to which the cell zone is attached. With this capability, a wide variety of problems that involve moving parts can be set up and solved using FLUENT. Depending on the level of complexity of the motion, and on the flow physics involved, one of FLUENT's moving cell zone models may be the most suited to your application.

### Overview of Moving Zone Approaches

The moving cell zone capability in FLUENT provides a powerful set of features for solving problems in which the domain or parts of the domain are in motion. Problems that can be addressed include the following

1. Flow in a (single) rotating frame
2. Flow in multiple rotating and/or translating reference frames

**Dynamic mesh:**

The dynamic mesh model in FLUENT can be used to model flows where the shape of the domain is changing with time due to motion on the domain boundaries. The motion can be a prescribed motion (e.g., you can specify the linear and angular velocities about the center of gravity of a solid body with time) or an unprescribed motion where the subsequent motion is determined based on the solution at the current time (e.g., the linear and angular velocities are calculated from the force balance on a solid body).

The update of the volume mesh is handled automatically by FLUENT at each time step based on the new positions of the boundaries. To use the dynamic mesh model, you need to provide a starting volume mesh and the description of the motion of any moving zones in the model. FLUENT allows you to describe the motion using either boundary problems or user-defined functions (UDFs).

FLUENT expects the description of the motion to be specified on either face or cell zones. If the model contains moving and non-moving regions, you need to identify these regions by grouping them into their respective face or cell zones in the starting volume mesh that you generate. Furthermore, regions that are deforming due to motion on their adjacent regions must also be grouped into separate zones in the starting volume mesh. The boundary between the various regions need not be conformal. You can use the non conformal or sliding interface capability in FLUENT to connect the various zones in the final model.

**Discretization**

Fluent uses a control volume based technique to convert the governing equations to algebraic equations that can be solved numerically. This control volume technique consists of integrating the governing equations about each control volume, yielding discrete equations that conserve each quantity on a control volume basis.

**Initializing the Solution**

As because solving is done by iterative method, user must provide FLUENT with an initial “guess” for the solution flow field. In many cases, one must take extra care to provide an initial solution that will allow the desired final solution to be attained.

There are two methods for initializing the solution:

- Initialize the entire flow field (in all cells).
- Patch values or functions for selected flow variables in selected cell zones or "registers" of cells.

## Convergence and Stability

### *Under Relaxation*

Because of the nonlinearity of the equation set being solved by FLUENT, it is necessary to control the change of $\phi$. This is typically achieved by under relaxation, which reduces the change of $\phi$ produced during each iteration. In a simple form, the new value of the variable $\phi$ within a cell depends upon the old value, $\phi_{old}$, the computed in $\phi$, $\Delta\phi$, and the under relaxation factor, $\alpha$, as follows :

$$\phi = \phi_{old} + \alpha\Delta\phi$$

By controlling relaxation factor one can avoid the sudden divergence in the solving process.

### *Monitoring Residuals*

During the solution process one can monitor the convergence dynamically by checking residuals, statistics, force values, surface integrals, and volume integrals.

### *Judging the convergence*

At the end of each iteration, the residual sum for each of the conserved variables is computed. On a computer with infinite precision, these residuals will go to zero as the solution converges. On an actual computer, the residuals decay to some small value ("round off") and then stop changing ("level out"). For "single precision" computations (the default for workstations and most computers), residuals can drop as many as six orders of magnitude before hitting round off. Double precision residuals can drop up to twelve orders of magnitude. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities such as drag or heat transfer coefficient.

### 3.4 PROBLEM SOLVING STEPS

After determining the important features of the problem following procedural steps are followed for solving it

1. Create the geometry model and mesh it.
2. Start the appropriate solver for 2D or 3D modeling.
3. Import the grid and check it.
4. Select the solver formulation
5. Chose the basic equation to solved: laminar or turbulent (or in viscid), chemical species or reaction, heat transfer models, etc. Also identify additional models needed: fans, heat exchangers, porous media, etc.
6. Specify the material properties.
7. Specify the boundary properties.
8. Adjust the solution control parameter.
9. Initialize the flow field.
10. Calculate a solution.
11. Examine the results.
12. Save the results.
13. If necessary, refine the grid or consider revisions to the numerical or physical model.

# CHAPTER-IV

## CFD ANALYSIS OF SCREW COMPRESSOR

Importance of screw compressor profile generation

Preparation of geometry and meshing

Computational domain and mesh to find out leakage flow rates

Moving reference frame

2-D model for rotation

3-D model of screw compressor

# CFD ANALYSIS OF SCREW COMPRESSOR

## 4.1 IMPORTANCE OF SCREW COMPRESSOR PROFILE GENERATION

One of the most important things is screw compressor profile generation because an efficient screw compressor needs a rotor profile which has large flow cross section area, short sealing line and small blowhole area. The larger the cross section area the higher the flow rate for same rotor sizes and rotor speeds. Shorter sealing line and smaller blow hole reduce leakages.

Higher flow and smaller leakage rates both increases compressor volumetric efficiency, which is the rate of flow delivered as a fraction of sum of the flow plus leakages. This in turn increases the adiabatic efficiency because less power is wasted in compression of gas which is recirculated internally.

There are different types of profiles available for screw compressors. For finding out the performance of any screw compressor in CFD using FLUENT software it is required that correct profiles in the GAMBIT are obtained. The present work has been done, by using profile points from DISCO (design integration for screw compressor optimization), a software which is developed by City University London.

To generate accurate profiles for the screw compresor, we need to import profile from DISCO to MASTER CAM, to analyze all points of the profile and write into text form or another procedure is to get the profile by using “converters” option in the MASTER CAM into IGES format. Generating a face to any further analysis in GAMBIT is done by joining all points with nurbs and making face with wire frame model. Some of the 2-D screw compressor profiles drawn in GAMBIT are shown below in figures in 4.1(a)-(d).

## 4.2 PREPARATION OF GEOMETRY AND MESHING

### 4.2.1 2-D Screw compressor rotor profiles

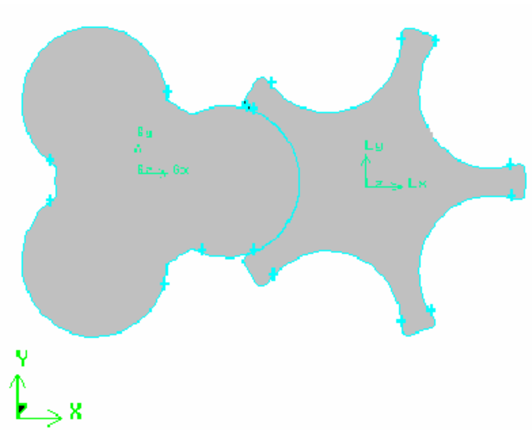


*Figure 4.1 (a): 3 by 5 symetric profile*

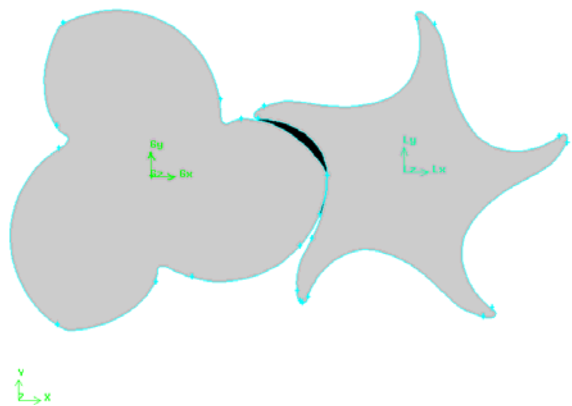


*Figure 4.1 ( b): 3 by 5 Asymmetric "N" profile*

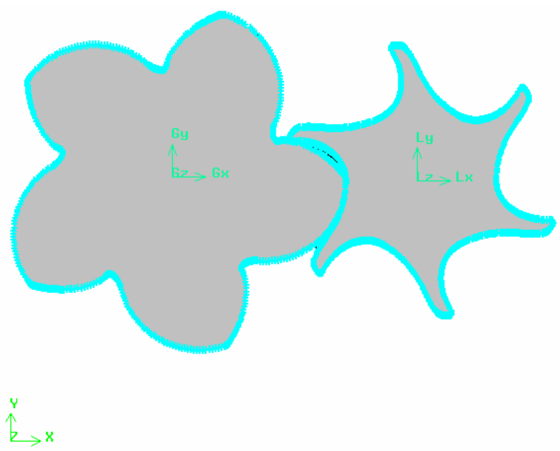


*Figure 4.1 (c) : 4 by 6 Asymmetric "N" profile*

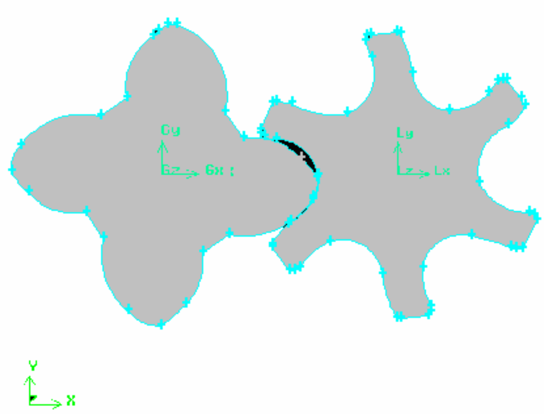


*Figure 4.1 (d) : 5 by 6 Asymmetric"N" profile*

Figure 4.1 (a) shows 3 by 5 symmetric profile which has less number of edges (16), while male consists of six edges and female consist of ten edges. It consists of circular arcs.If numbers of edges are less, it has the advantage that meshing takes less time.

**The design specification of the compressor is as follows**:

| | |
|---|---|
| Type: | 101.5 mm rotor diameter, 3/5 oil dry screw compressor. |
| L/D | 1.1 |

***A. General*** 3/5-101.5 mm outer diameter, 111.650 mm length

***B. Profile***

"N" Rack Generated Profile, UK Patent 9610289.2, PCT World 97/43550, US 6,296,461, EP 0 898 655

| | |
|---|---|
| Lobe combination | 3/5 |
| Volume constant [Cdp] | 0.511 |
| Overlap constant [Col] | 0.960 (290°) |

***C. Basic Geometry***

| | |
|---|---|
| Centre distance | 74.000 REF |
| Pitch dia, male rotor | 55.500 REF |
| Pitch dia, female rotor | 92.500 REF |
| Rotor dia, male rotor | 101.500 +0.000/0.000 |
| Root dia, male rotor | 52.300 +0.000/0.000 |
| Rotor dia, female rotor | 95.300 +0.000/0.000 |
| Root dia, female rotor | 46.100 +0.000/0.000 |
| Rotor length | 111.650 +0.000/-0.008 |
| Length/dia ratio | 1.10 |
| Wrap angle | 249° |
| Lead, male rotor | 161.640 RH helix |
| Lead, female rotor | 269.400 LH helix |
| Helix angle at pitch | 47.1678° |
| Lead angle at pitch | 42.8323° |
| Displacement | 0.59 lit/rev |

Figure 4.1(b) shows 3 by 5 Asymmetric " N "profile which has less number of edges(22), while male consists of nine edges and female consist of ten edges. Profile consists of different curves as shown in figure 4.3.

Reduced centre distance Reduced A mm

Backlash μm 7-84

| | Point | Clearance |
|---|---|---|
| Clearances μm | A-B | 200-200 |
| | C | 200-200 |
| | D | 165-165 |
| | E | 165-165 |
| | F | 200-200 |

| | |
|---|---|
| G | 200-200 |
| H | 200-200 |
| I-A | 200-200 |

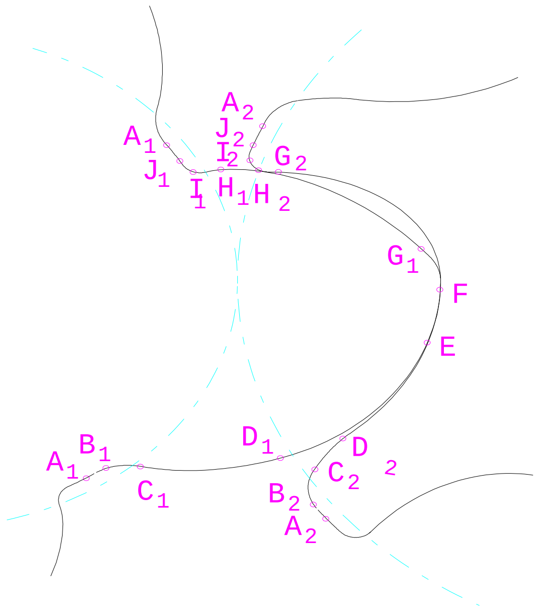



*Figure 4.2: clearances for the " N" asymmetric profile*

Figure 4.1(c) shows that profile drawn in AutoCAD which consists of more edges. These edges are drawn manually using the option "polylines" and it gives improper curves which is an inappropriate profile. This procedure can use any type of curve which does not know about original coordinates and is exported to GAMBIT as ACIS format.

The profile shown in Figure 4.1 (d) consists of 1072 points input to the preprocessor as a text form and made face by "polygon" option .This face consists of 1072 edges and takes more time to make volume and mesh generation.

N Rotor Profile:

E-F Circle

F-G Straight Line

G-H Undercut by the Gate Rotor

H-A Undercut by the Main Rotor

A-B Arc: P=0.43,q=1

B-C Straight Line

C-D Circle

D-E Straight Line

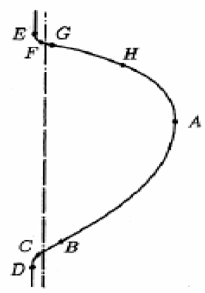



*Figure 4.3: different arcs for "N" asymmetric profile*

### 4.2.2 Different 3D rotors

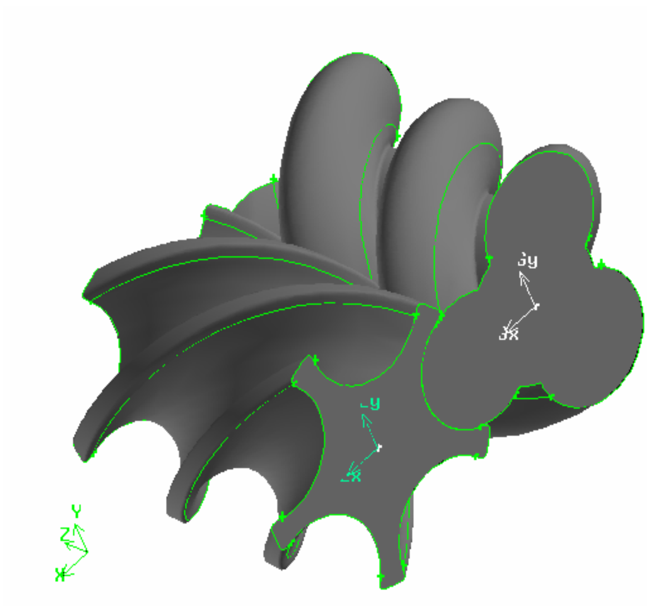



*Figure 4.4 (a): 3 by 5 symmetric rotors*


*Figure 4.4 (b): 3 by 5 Asymmetric rotors*

*Figure 4.4 (c): 4 by 6 asymmetric rotors*

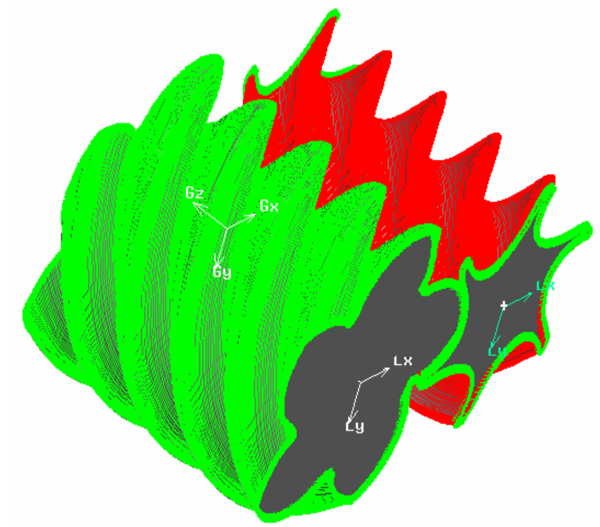



*Figure 4.4 (d): 5 by 6 asymmetric rotors*

Figure 4.4(a) It consists of 3 by 5 symmetric rotors generated by sweeping their faces along their axial direction with their wrap angles.

Male rotor wrap angle 248.664°

Female rotor wrap angle 149.198°

Length to diameter ratio 1.1

Figure4.4 (b) It consists of 3 by 5 Asymmetric rotors generated by sweeping their faces along their axial direction with their wrap angles.

Male rotor wrap angle 248.664°

Female rotor wrap angle 149.198°

Length to diameter ratio 1.1

Figure 4.4(c) It consists of 4 by 6 symmetric rotors generated by sweeping their faces along their axial direction with their wrap angles.

Male rotor wrap angle 300°

Female rotor wrap angle 200°

Length to diameter ratio 1.5

Figure 4.4 (d) It consists of 5 by 6 symmetric rotors generated by sweeping their faces along their axial direction with their wrap angles.

Male rotor wrap angle 300°

Female rotor wrap angle 200°

Length to diameter ratio 1.1

The compression process depends on the male wrap angle. This angle gives the angle of contact between male and female rotors during compression and determines the axial movement of compression process. A smaller wrap angle gives higher speed of compression, where as for larger male wrap angle ,speed of compression is lower for same volume delivery. Different combinations of screw compressor rotors have different wrap angles.

### 4.2.3 Different views of 3 by 5 screw compressor :

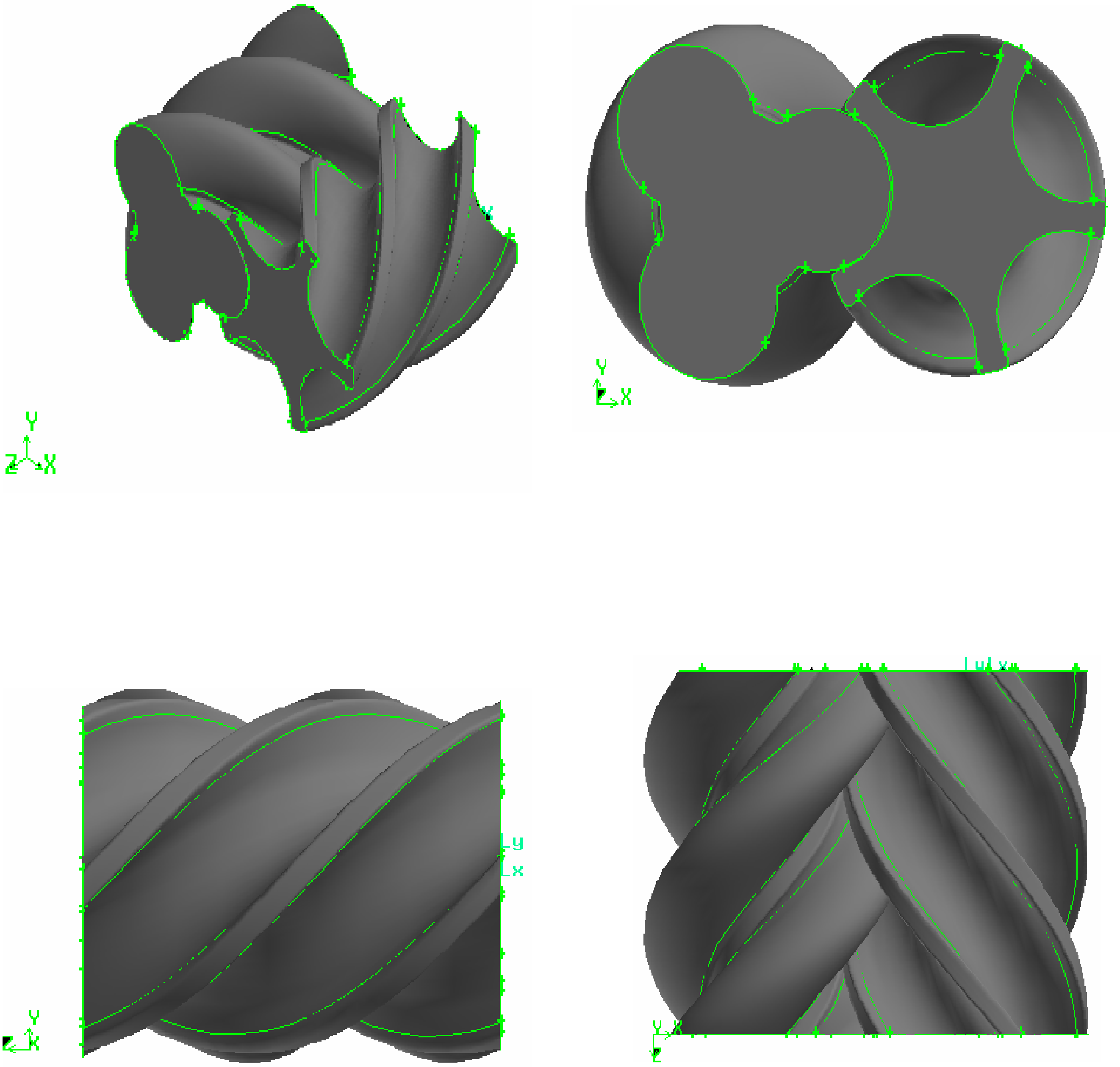


*Figure 4.5: Different views of 3 by 5 screw compressor*

### 4.2.4 Casing:

Casing face is created by producing circular faces around rotor profiles with a minimum clearance and uniting two faces creating an intersection of two circular face in 2D.the casing volume is generated by sweeping along the axial direction with a slight excess length of rotors in 3D and moving the casing to negative side in such a way that its acts as end plates of compressor

Make inlet and outlet ports by creating cylinders of appropriate dimensions. Locate the cylinders at different and appropriate locations. Unite the three volumes-casing, body and ports to get entire casing part.

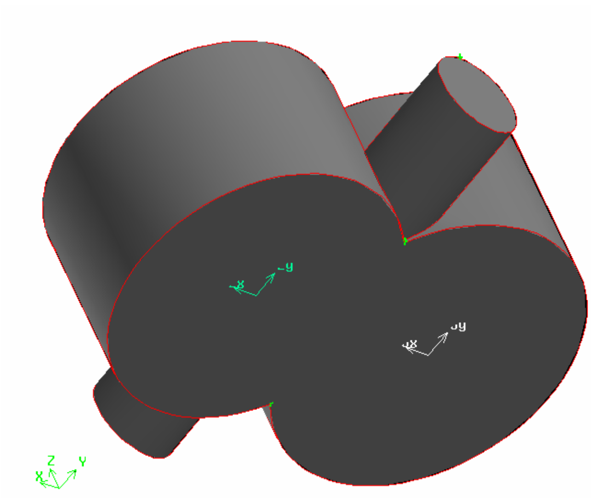


*Figure 4.6 (a): casing modify model*

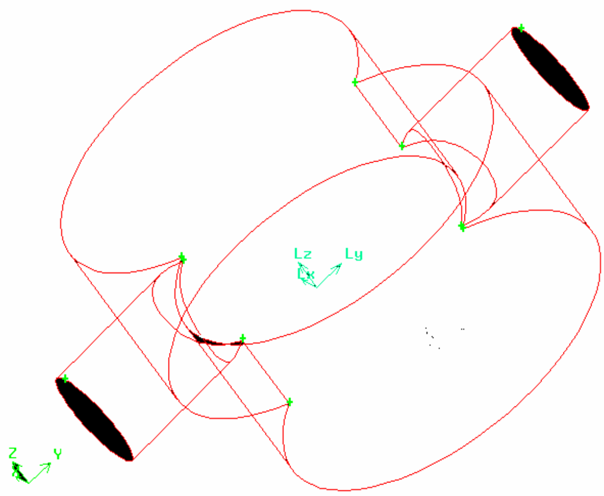


*Figure 4.6 (b): wire frame of casing*

While generating the casing, it should be taken care that male and female should not come outside the casing (otherwise it will show negative volume-this being an error). The error can minimize by choosing proper clearance values.

### 4.2.5 Final model to FLUENT

By subtracting two volumes, male and female, from casing we get a domain to procced further analysis. The modified shape of casing with rotors and wire frame model casing with rotors are shown in figure 4.7(a) and 4.7 (b).

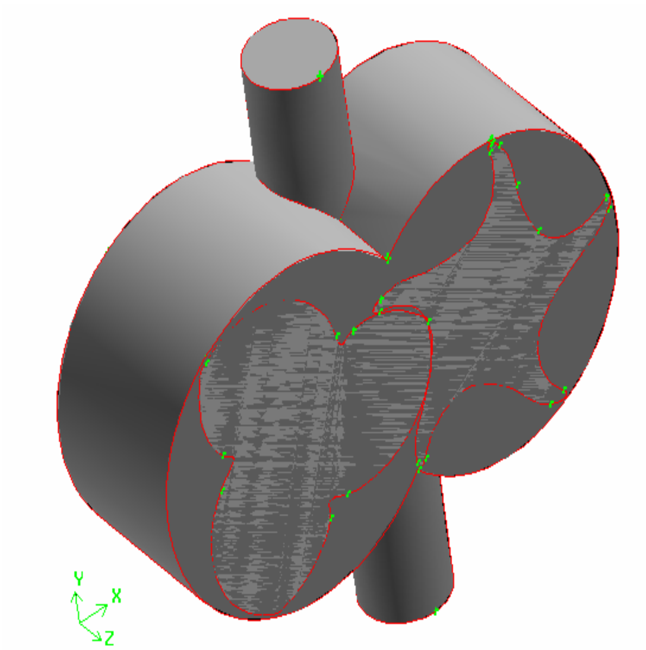



*Figure 4.7 (a): modified shape of casing with rotors*

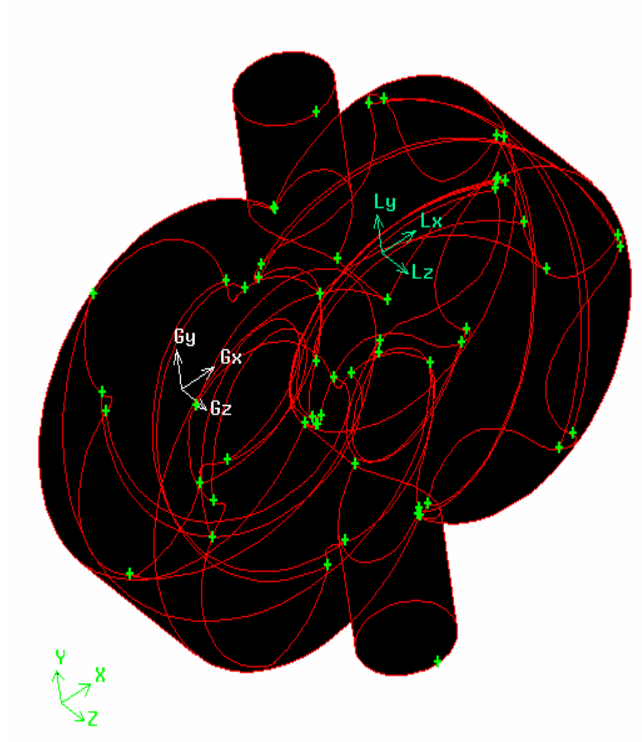



*Figure 4.7(b): wire frame model casing with rotors*

### 4.2.6 Meshing:

Meshing is most important part of GAMBIT and to that of FLUENT. There are different types of mesh schemes available in the preprocessor. But generating mess with less number of grids and less skewness is highly challenging. Once the geometry is prepared, the next step is to mesh the volume. In this step, the volume is subdivided into small number of volumes.

Generation of structured grid for any geometry is preferred. Meshing is most important part for FLUENT, that influences the accuracy of the problem. For screw compressor type of geometry, it is difficult to generate structured grid. For this structured grid has been generated with minimum skew ness in comparison to the default schemes .

**Male:**

In the above geometry is generated by making a circular face around the male face and then making "split" male face from the circular face and "spliting" that total face into three faces. Start from edge mesh and mesh the face.

Total number of faces ……Three

Total nodes…….. 1926

Total elements …… 1590

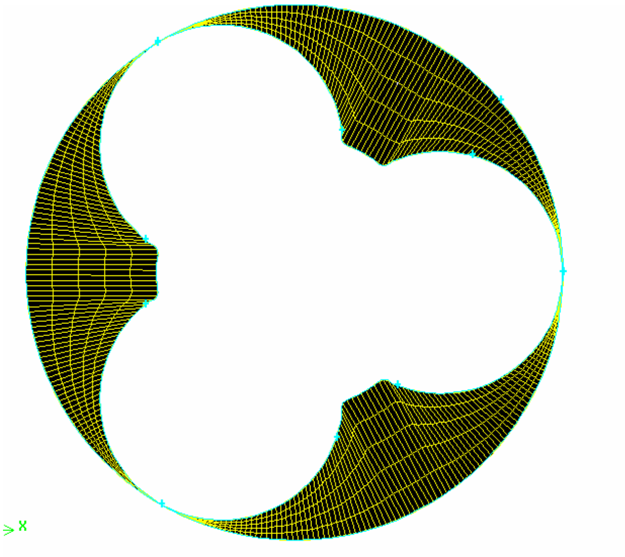


*Figure 4.8 (a): Working domain of male rotor*

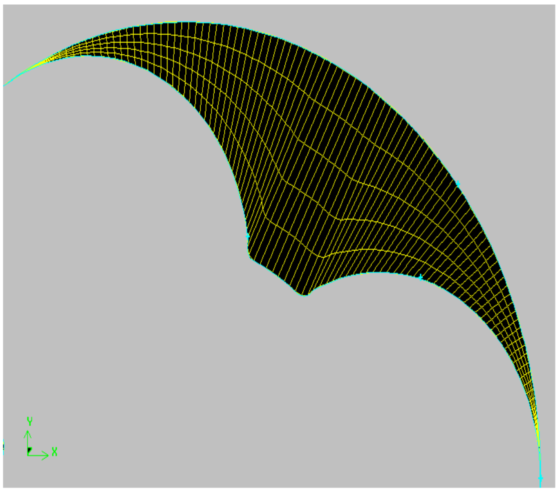


*Figure 4.8 (b): Cross section of male casing*

## Female

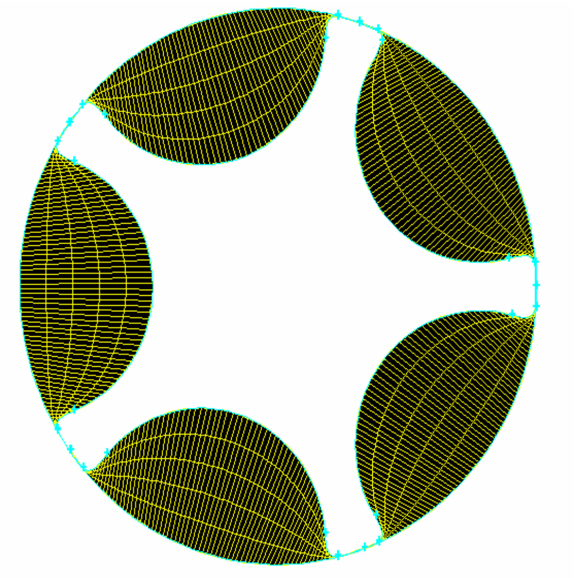


*Figure 4.9(a) : Working domain of  female rotor*

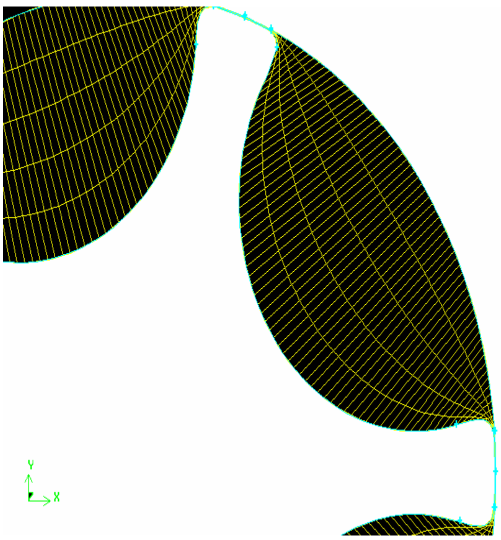


*Figure 4.9 (b): Cross section of female casing*

**Female meshing**

Total faces …… ………….10

Total nodes…….. 326

Total elements …… 1220

The rotors of a screw compressor are helical type elements generated by the simultaneous revolution of the rotor profile around the rotor axis and its translation along the axis. The entire screw compressor geometry can thereby be generated within a two-dimensional definition of the rotors by the calculation of points in cross sections and connecting them with appropriate vertices in other cross sections.

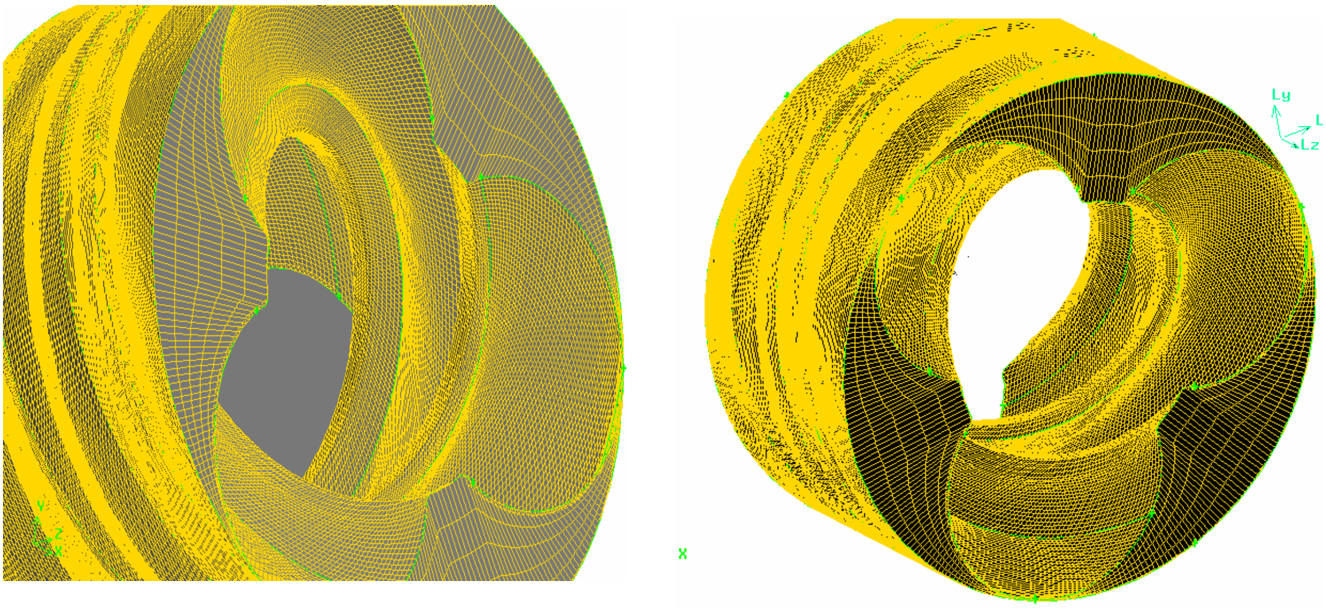


*Figure 4.10: Sweeping of face meshes with along axial direction*

**Rotors meshing**

Figures 4.11(a),(b) show male and female solid rotors which are meshed by scheme Tet\Hybrid elements and type T grid .Number of cells for male and female rotors are 18, 98,551 and 9,76,275 respectively

Figures 4.11(c),(d) show male and female solid rotors which are meshed by scheme Tet\Hybrid elements and type T grid .Number of cells for male and female rotors are 15,03,418 and 8,47,356respectively by this reduction cells are more.

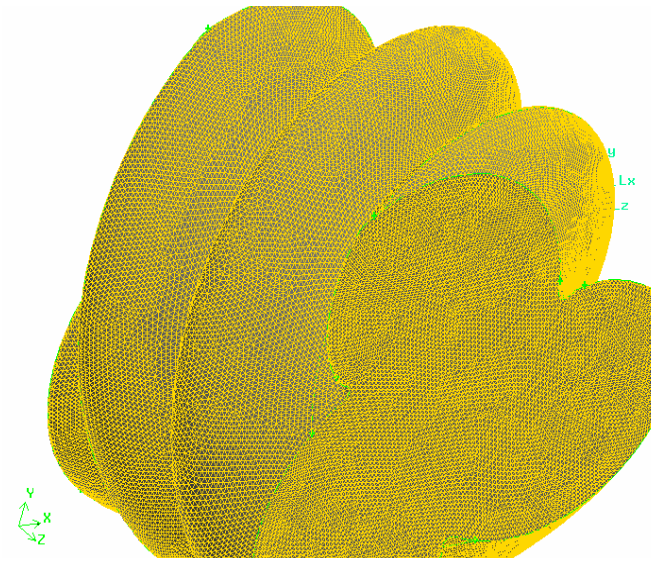


*Figure 4.11 (a): Meshing of male rotor with T grid*

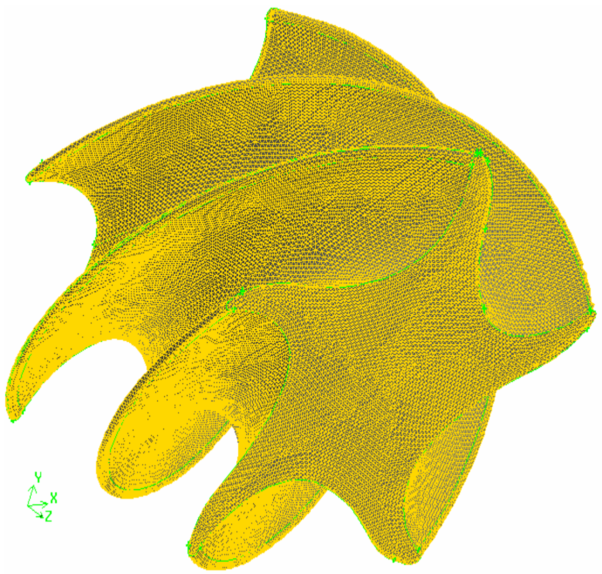


*Figure 4.11 (a): meshing of female rotor with T grid*

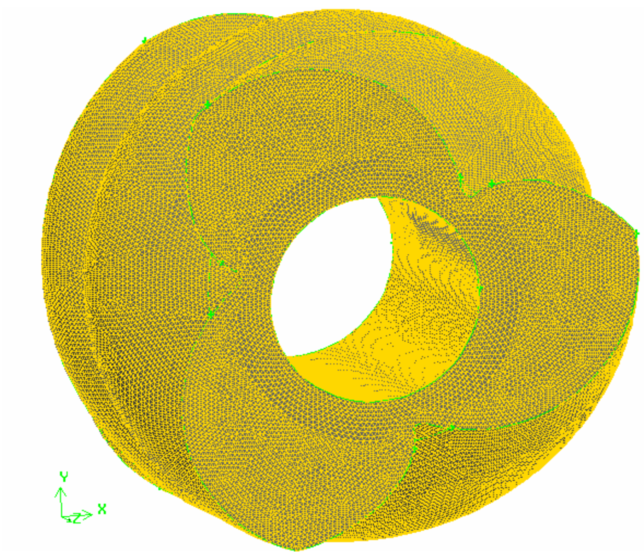


*Figure 4.11(c): Meshing of Male rotor by subtracting inside rotor*

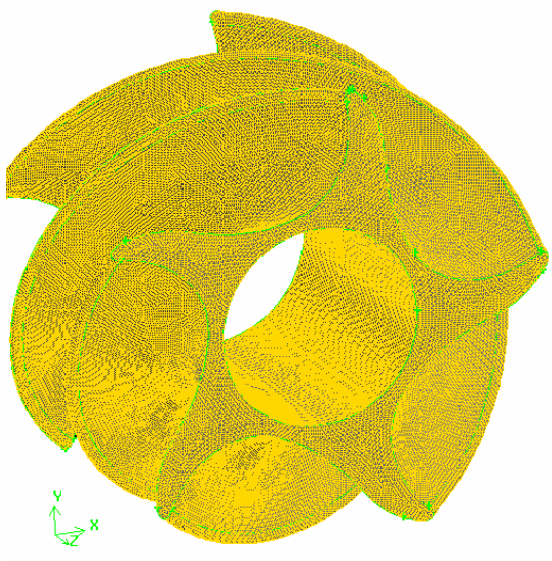


*Figure 4.11(d): Meshing of Female rotor by subtracting inside rotor*

## 4.3 COMPUTATIONAL DOMAIN AND MESH TO FIND OUT LEAKAGE FLOW RATES

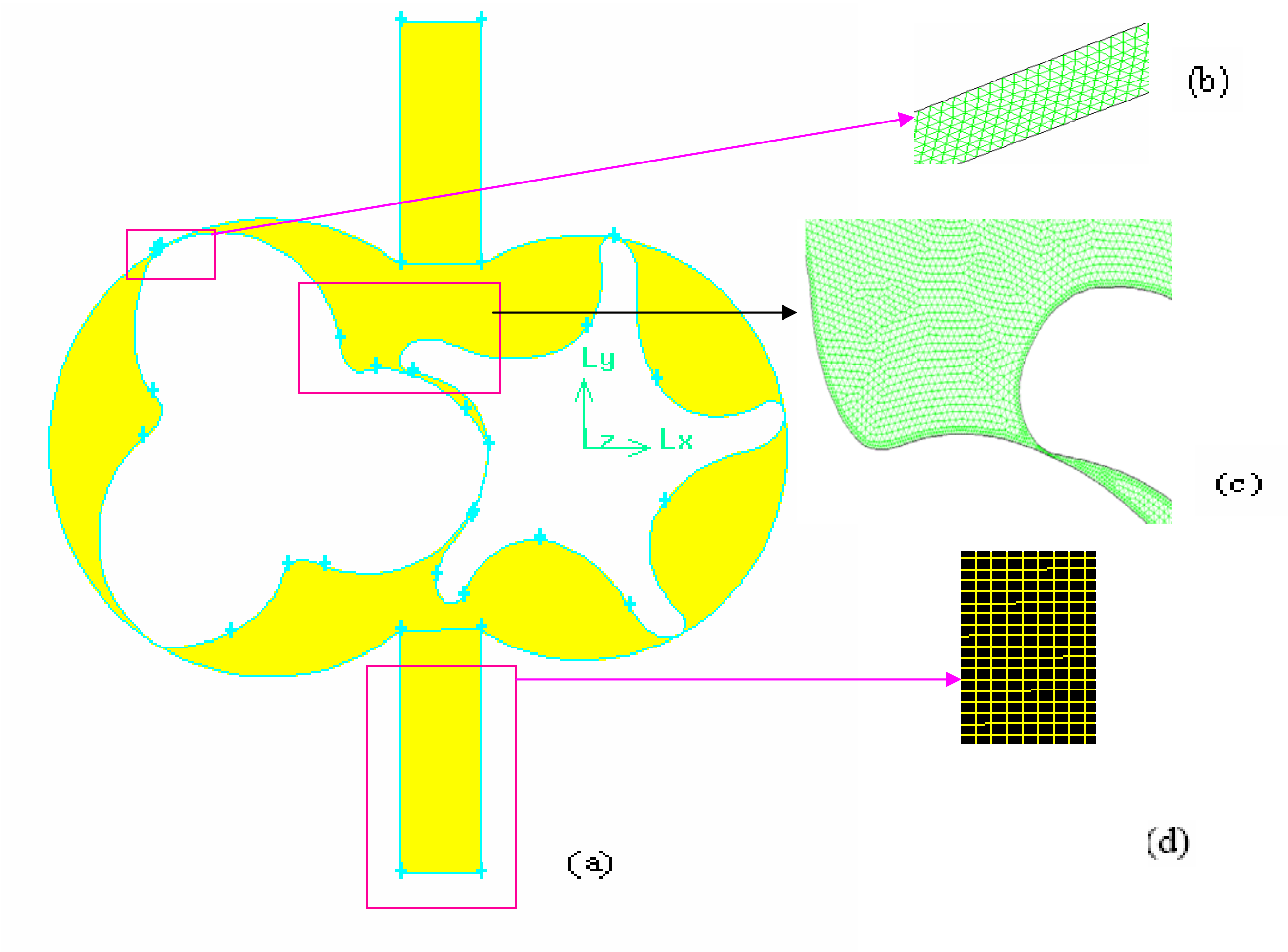



*Figure 4.12: A meshed two-dimensional model of screw compressor*

Figure 4.12(a) shows the two-dimensional model of a 3 by 5 screw compressor, which contains two rotors, casing, inlet and outlet. The flow area is meshed in different grid species in order to get more accurate simulation results and less computational time. As shown in Figure 4.12(b), the small clearance between the rotors and casing is meshed by very fine grid, since this area is the key part of the whole domain. An unstructured mesh arrangement with triangle elements, as shown in Figure 4.12(c), is applied to the domain, which is around the rotors, because it contains curves and corners. The locations of the inlet and outlet are placed far enough in order not to affect the flow in the casing. These two areas are created by a regular, structured grid of quadrilateral elements.

## 4.4 MOVING REFERENCE FRAME

### 4.4.1 Flow through Male and Female Flute:

Figure 4.13 shows 4/6 profile male flute. The 4/6 profile generated from AutoCAD profile is imported to Gambit in ACIS format. A centre for the imported male & female is created to draw a circle with that centre. The male rotor in 2-D is “split”. Volume is generated by sweeping the face along axial direction with 1.5 times of outer circular diameter and twist angle 300.The outer diameters of male flute and female flute are38 cm 30 cm respectively.

### Male flute meshing:

This flute is meshed under Tet/hybrid scheme with T grids .Number of cells of the male flute 24780 volumes

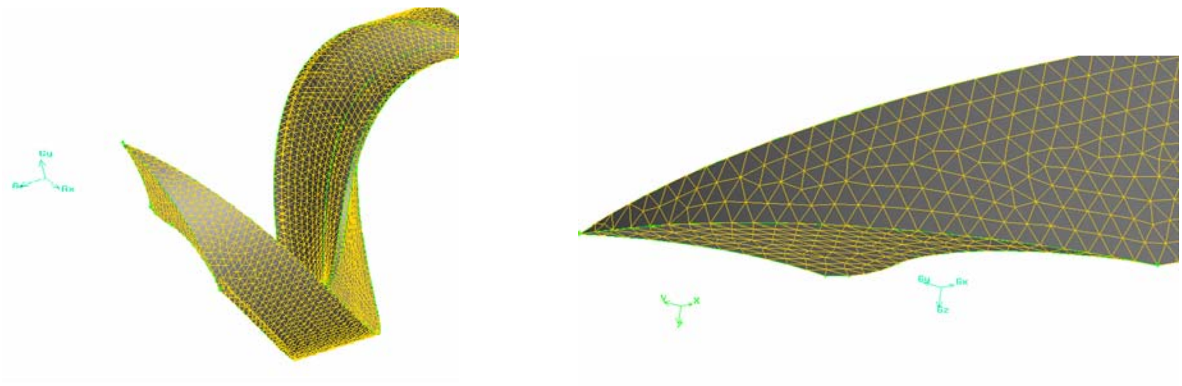


*Figure 4.13: Model of male flute in the GAMBIT*

### Flow analyses in fluent:

Male Flute consists of seven faces in which four are side faces, top face of casing, inlet and outlet. Boundary conditions are: pressure inlet and pressure out let and side and top faces are rotating.


***Figure 4.14 :*** *Velocity Contours of Male Flute*

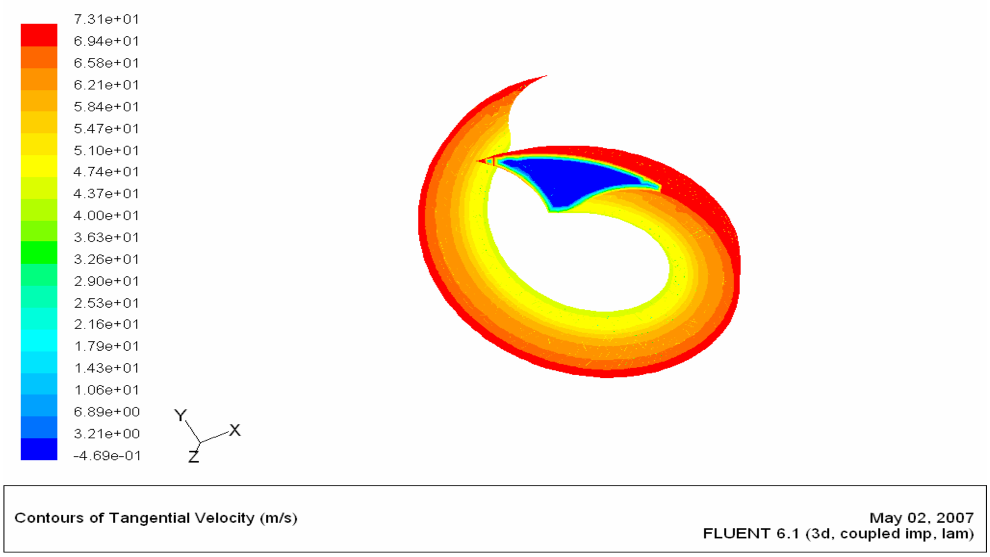



*Figure 4.15 : Tangential velocity contours of Male Flute*

**Female Flute:**

Female Flute consists of seven faces in which four are side faces one top face of casing, one inlet and one outlet. Boundary conditions are: pressure inlet and pressure out let and side and top faces are rotating just like the male flute.

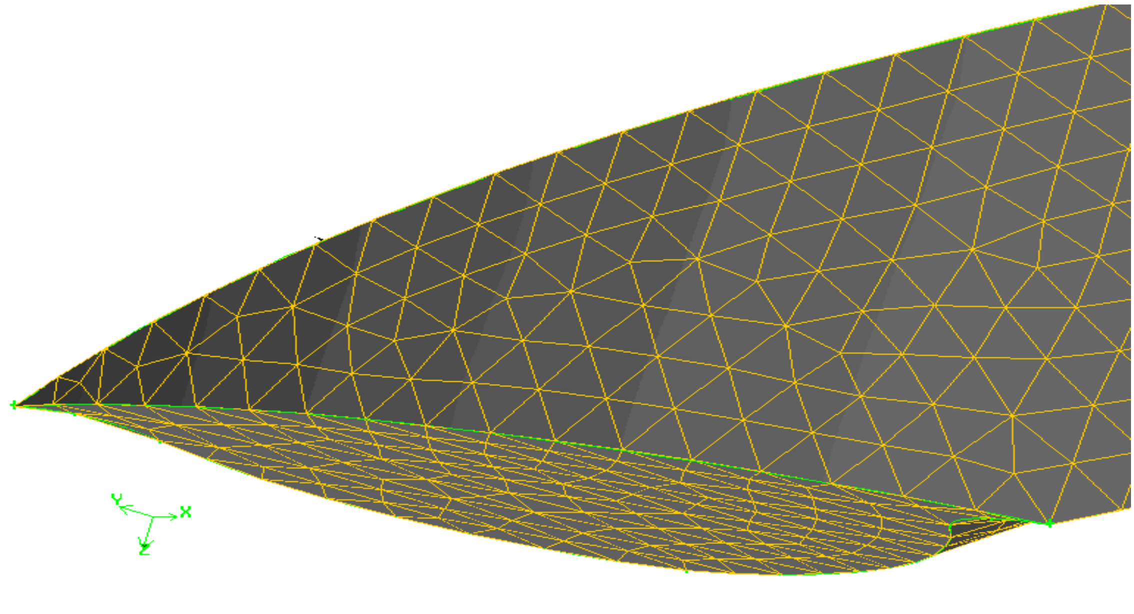


*Figure 4.16: Cross section view of female flute*

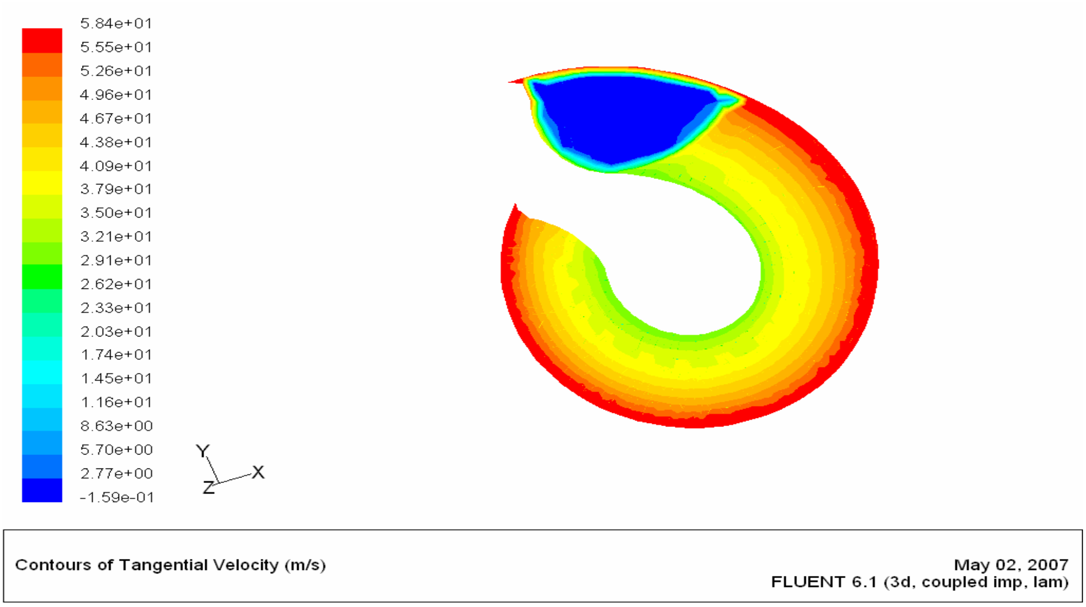



*Figure 4.17 Tangential velocity contours of female flute*

## 4.5 2-D MODEL FOR ROTATION:

Slight modification of figure 4.12 is that the boundary conditions of the edges of male rotor are earmarked as Male "WALL" and the boundary conditions of the edges of female rotor are earmarked as Female "WALL". Now Male and Female WALLS are separately mentioned GAMBIT.

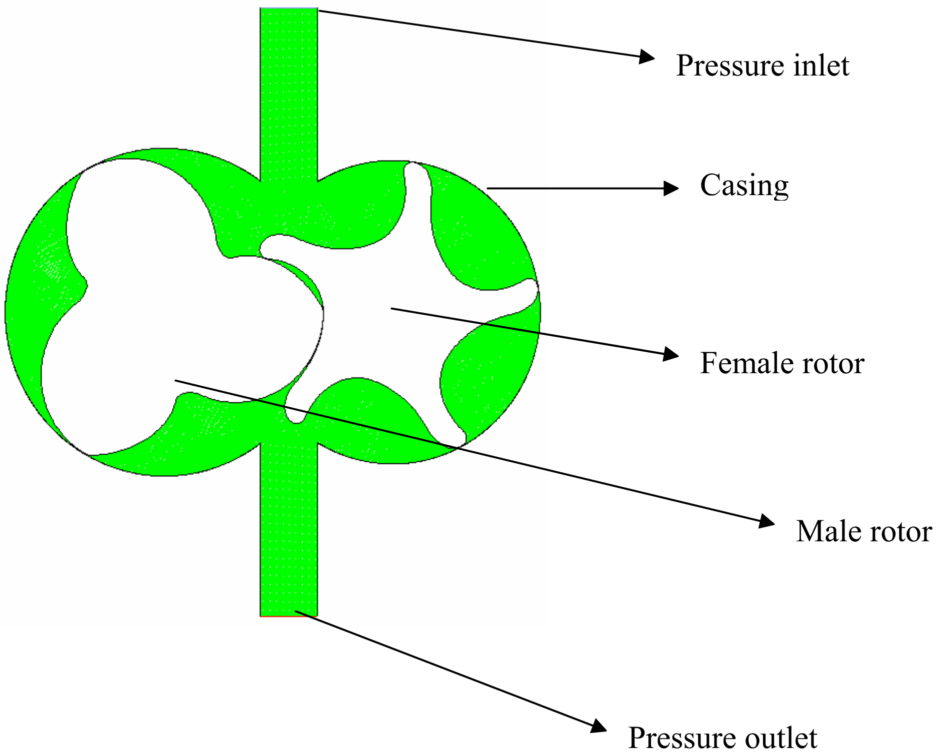



*Figure 4.18: computational domain for 2-D*

## 4.6 3-D MODEL OF SCREW COMPRESSOR:

Generation of 3-D screw compressor working domain by starting from lower geometry as following procedure:

**Preprocessing:**

1 Generation of faces of screw compressor rotors

2 Make volumes by giving twist and sweep along the axial direction of their respective centers

3 Casing is generated by creating circular faces around the rotors with minimum clearances, unite those two faces and make total face to sweep along axial direction.

4 Make two ports by using cylinder option.

5 By making uniting ports and casing we can get entire casing with inlet and outlet port

6 By subtracting two rotors from the casing we can get screw compressor working domain

**Meshing:** By giving default scheme Tet\hybrid with T grid an interval size of 1.2 and generated 19, 22,153 cell in the entire compressor.

**Boundary conditions:**

Inlet and outlet conditions are pressure inlet and pressure outlet respectively, and given faces inside compressor are male and female respective manner.

**FLUENT:**

For the present problem we have used Segregated solver with turbulence K- ϵ model and have given 200 Pascal pressure inlet and turbulent and viscosity ratio are 2% and 10.

**Flow through static rotors:**

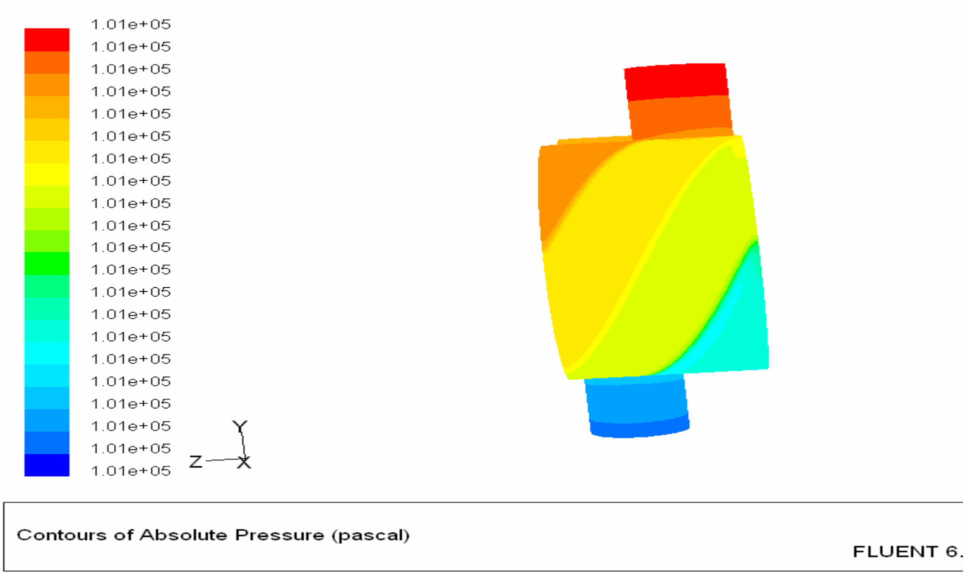



*Figure 4.19: Pressure contours of screw compressor*

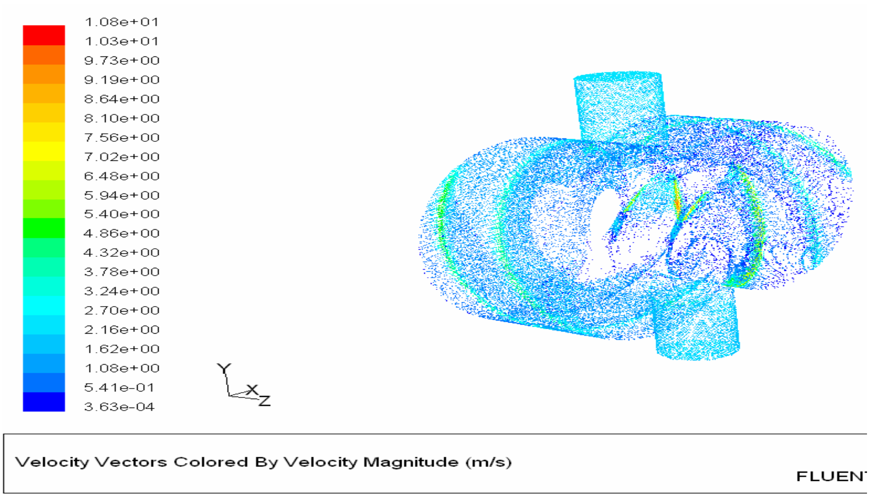



*Figure 4.20: velocity vectors through screw compressor*

# CHAPTER-V

## RESULTS AND DISCUSSION

CFD model verification

Flow behaviors and results analysis in two-dimensional models

# RESULTS AND DISCUSSION

In this study, a number of CFD models, including 2D and 3D, have been developed to analyses flow through twin-screw compressor. By using these models, the leakage flow rate through each leakage pathway has been quantified.

## 5.1 CFD MODEL VERIFICATION

In this investigation, it is assumed that the leakage is independent of the flow induced by the rotor motion and that the leakage can be assessed by a steady flow at a series of rotor positions with a given pressure drop. Although FLUENT provides reasonable accuracy of the physical model, it still needs experimental results to validate the numerical results from this model. Apart from Fleming and Tang [64], there is hardly any reported research on leakage in twin-screw compressors.

## 5.2 FLOW BEHAVIORS AND RESULTS ANALYSIS IN TWO-DIMENSIONAL MODELS

Figure 5.1 shows the pressure contours and velocity vectors of a two-dimensional model for a 3 by 5 screw compressor.

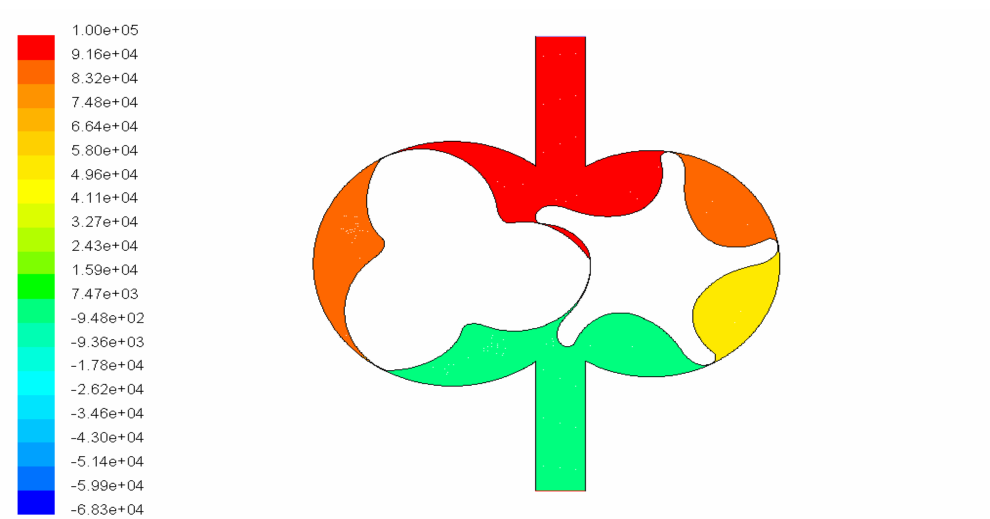



*Figure 5:1 Contours of Static Pressure (Pascal)*

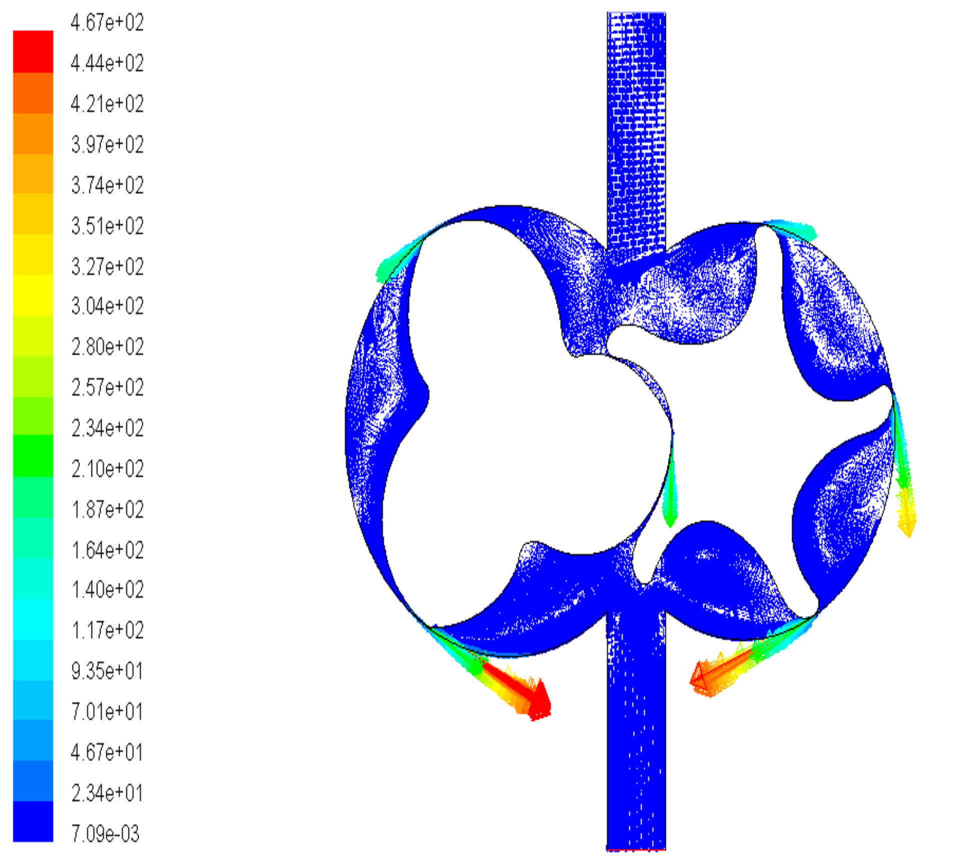



*Figure 5.2: Velocity Vectors Colored By Velocity Magnitude (m/s)*

It can be seen clearly from Figure 4.13 that the pressure is high in the discharge area on the top, and low in the suction area on the bottom. Due to the pressure gradient between the two areas, the leakage flow passes through the clearance, which is between two rotors and rotor-casings, and returns to the suction area. Figure 5.2 presents that the highest velocity is between the male rotor and casing, which means the highest leakage rate, is in this domain.

In this table 5.1 leakage flow rates calculated for different pressure ratios are tabulated. Plots are drawn form the reading of the table. This is shown if Figure. 5.3. As expected all clearance flow rates increase with pressure ratio, but not in a linear fashion. The reason for that is that as the pressure ratio increases the air speed in the different clearance passages increases considerably. The friction losses are proportional to the square of the speed thus the non-linear behavior in Figure 5.3.

*Table 51: Leakage flow rates through each leakage pathway in the screw compressor*

| S.No | Pressure ratio | Leakage flow rate (kg/s) | | | |
|---|---|---|---|---|---|
| | | Male and casing | Female and casing | Male and female | Total |
| 1 | 1.5 | 0.044845 | 0.039239 | 0.014471 | 0.098556 |
| 2 | 2 | 0.053449 | 0.040412 | 0.018683 | 0.112543 |
| 3 | 2.5 | 0.073239 | 0.050502 | 0.020241 | 0.144004 |
| 4 | 3 | 0.087703 | 0.060617 | 0.024212 | 0.172532 |
| 5 | 3.5 | 0.092411 | 0.074938 | 0.024343 | 0.191691 |

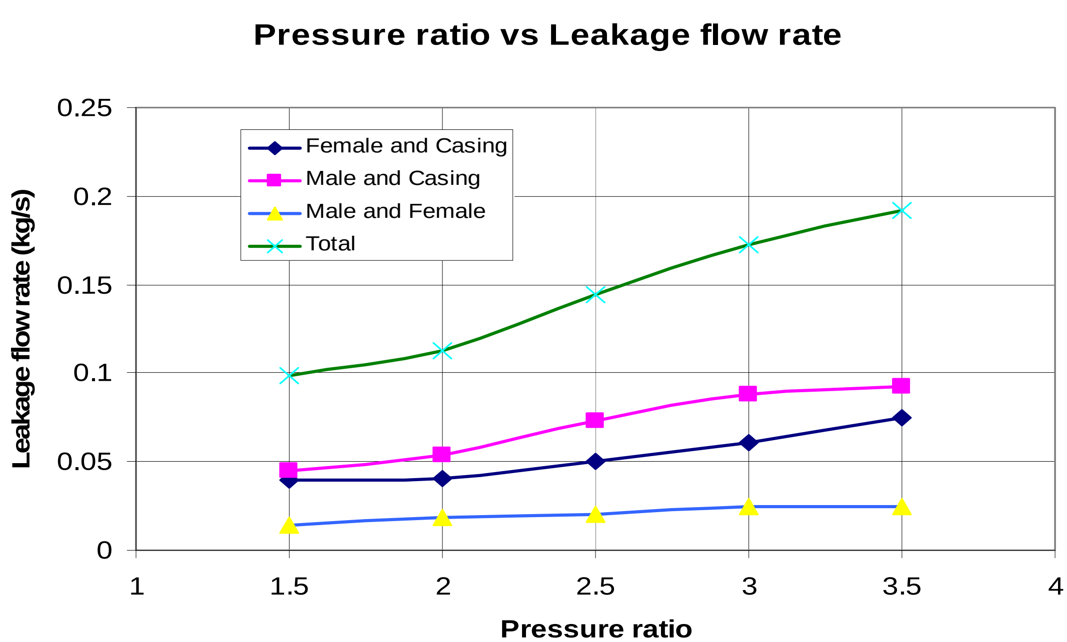



*Figure 5.3: Clearance flow rate as a function of pressure ratio. Rotors at "reference" position.*

The major clearance flow is between the male rotor and the casing (M-C) followed by that between the female rotor and the casing (F-C) and lastly between the female and male rotors (F-M). The cause of this can be traced to the flow resistance of each path, which is proportional to the length and width of each path. The overall trend in flow distribution between the three clearance paths does not change with pressure ratio.

Moving reference has been frame successfully applied for male, female flute analysis. For male and female flutes are getting maximum tangential velocities at the outer domain are 73.1 and 58.4 m/s respectively.

In the FLUENT boundary conditions for male and female WALLS have rotational speed of 3600 Rpm and -3600 Rpm respectively with their respective centers and using dynamic mesh for casing (which is a working medium) by "Remeshing" method and k-ϵ Turbulence model used for this model. These rotors are generated maximum pressure of 3.75 bar. Figure 5.4(a) shows how the velocities are changing in between the rotors and around the circumference of rotor.

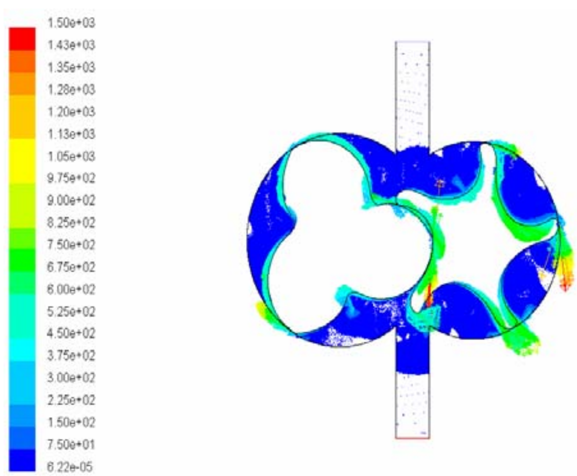



*Figure 5.4 (a): Velocity vectors while rotating screw compressor (m/s)*

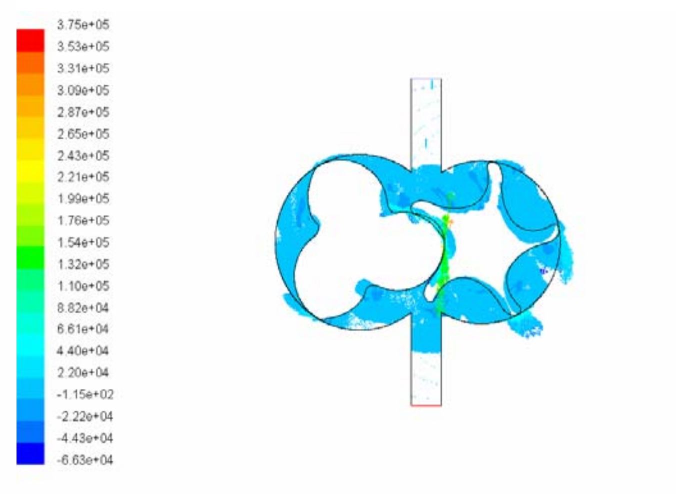



*Figure5.4 (b): Pressure vectors (pascal)*

Different types of profiles are generated. Simulation of 2-D and 3-D are created by using FLUENT.

# CONCLUSIONS

The performance of process gas screw compressors is highly dependent on their rotor profiles and clearance distribution.

While generation of Asymmetric “N” profiles consists of more curves care should be that while generation of faces. If more the number edges to make screw compressor profile number of faces are more to make 3-D geometry. We have to reduce number of edges such way that optimize the geometry.

Clearance is most judging parameter for performance of screw compressor it o . This enables the clearances between the rotors to be kept below 15μm. With such small clearances, rotor contact is very likely and, hence, the profile and its clearance distribution must be generated in such a manner that damage or seizure will be avoided should this occur.

The clearance flow rate is non-linearly proportional to the pressure ratio across the rotor casing assembly. Most of the flow goes through the M-C path followed by the F-C and F-M paths, respectively. We have found out pressure and velocity variation around the rotors.

## Future scope:

1. CFD studies on flow through oil injected screw compressor
2. Generation of structured grids for screw compressor is more challenging one.
3. Validation of the CFD models with experimental work

# References


[1] **Arbon, I. M.** The Design and Application of Rotary Twin-Shaft Compressors in the Oil and the Gas Process Industry, Mechanical Engineering Publication Limited, London (1994)

[2] **Brown, R. N.** Compressor: Selection and Sizing, Gulf Publishing Company Book Division, (1986)

[3] **Taft, G.L.** Selection and Application of the Industrial Screw Compressors, Proceedings of the Purdue Compressor Technology Conference, Purdue, USA (1972), 306-310.

[4] www.arielcorp.com

[5] **Stosic, N., Smith, I.K., Kovacevic, A.** Optimization of Screw Compressor Design, International Compressor Conference, Purdue, USA (2002), C21-23, 1177-1195. 71

[6] **Singh, P.J., Bowman, J.L.** Effect of Design Parameters on Oil Flooded Screw Compressor Performance, Proceedings of the International Compressor Engineering Conference, Purdue, USA (1986), Vol.I, 71-88

[7] **Stosic, N.** Screw Compressors: Future Developments, Published on Internet; (1999) (www.city.ac.uk/~sj376).

[8] **Stosic, N., Smith, I.K., Kovacevic, A.** Opportunities for Innovation with Screw Compressors, (www.city-compressors.co.uk)

[9] **Stosic, N., Smith, I., Kovacevic, A.** Screw Compressors: Mathematical Modelling and Performance Calculation, Springer (2005)

[10] **Tang, Y., Fleming, J.S.** Obtaining the Optimum Geometrical Parameters of Refrigeration Helical Screw Compressor, Proceedings of the International Compressor Engineering Conference, Purdue, USA (1992), Vol.I, 221-227

[11] **Sjoholm, L.** Important Parameters for Small Twin Screw Refrigeration Compressors, Proceedings of the International Compressor Engineering Conference, Purdue, USA (1986), Vol.I, 1129-1144

[12] **Singh, P.J., Schwartz, J.R.** Exact Analytical Representation of Screw Compressor Rotor Geometry, International Compressor Engineering Conference Purdue, USA (1990), Vol.I, 925-937

[13] **You, C.X., Tang, Y., Fleming, J.S.** Optimum Rotor Geometrical Parameters in Refrigeration Helical Twin Screw Compressor, International Compressor Engineering Conference, Purdue, USA (1996), Vol.I, 1-6

[14] **Stosic, N., Hanjalic, K.** General Method for Screw Compressor Profile Generation, International Compressor Engineering Conference, Purdue, USA (1996), Vol.I, 57-162

[15] **Stosic, N., Hanjalic, K.** Development and Optimization of Screw Machines With a Simulation Model-Part I, Profile Generation, Journal of Fluid Engineering, (1997), **119**, 659-662

[16] **Stosic, N., Hanjalic, K.** Development and Optimization of Screw Engine Rotor Pairs on the Basis of Computer Modelling, Purdue Engineering Conference, Purdue, USA (1994), Vol.I, 55-60

[17] **Stosic, N., Smith, I K., Kovacevic, A.** Rotor Interference as a Criterion for Screw Compressor Design, J. Engg. Design, Vol.14, **2** June (2003), 209–220

[18] **Singh, P.J., Onuschak, A.D.** A Comprehensive, Computerized Method for Twin Screw Rotor Profile Generation and Analysis, Proceedings of the Purdue Compressor Technology Conference, Purdue, USA (1984), 519-527

[19] **Mccreath, P., Stosic, N., Smith, I.K., Kovacevic, A.** The Design of Efficient Screw Compressors for Delivery of Dry Air, International Conference, Compressors and their Systems, London, (2001)

[20] **Venumadhav, K., Stosic, N., Smith, I.K., Kovacevic, A.** The Design of a family of Screw Compressors for Oil Flooded Operation, (www.city.ac.uk/~sj376)

[21] **Stosic, N., Smith, I.K., Kovacevic, A.** Optimization of Screw Compressors, Applied Thermal Engineering, 23 (2003) 1177-1195

[22] **Sjoholm, L.I., Muralidhar, A.** Measurement of the Discharge Port for a Screw Compressor, Fifteenth International Compressor Engineering Conference, Purdue, USA (2000), 995-1002

[23] **Xiong, Z.** The Dynamic Measurement and Mating Design of a Screw Compressor Rotor Pair, Proceedings of the International Compressor Engineering Conference, Purdue, USA (1984), 554-558

[24] **Fujiwara, M., Kasuya, K., Matsunaga, T., Makoto, W.** Computer Modelling for Performance Analysis of Rotary Screw Compressor, Proceedings of the Purdue Compressor Technology Conference, Purdue, USA (1984), 536-543

[25] **Abu-Hijlleh, B., Tu, J., Subic, A., Li, H., Ilie, K.** CFD Comparison of Clearance Flow in a Rotor-Casing Assembly using Asymmetric Vs Symmetric Lobe Rotors, American Society of Mechanical Engineers, Fluids Engineering Division (FED), Vol.259, (2003), 327-330

[26] **Ahmed Kovacevic.,** Boundary Adaptation in Grid Generation for CFD Analysis of Screw Compressors

[27] **Ahmed Kovacevic, Nikola Stosic, Ian K. Smith** numerical simulation of fluid flow and solid structure in screw compressors

[28] **H.F. Li, J.Y. Tu, and A. Subic.**The Three-Dimensional CFD Analysis of Flow Behaviors in a Complex Twin Screw Supercharger

[29] **N.Stosic I Smith A Kovacevic**, Screw Compresors Mathematical Modelling and performance Caluclation.

[30] **Seshaiah,N.,Ghosh,S.K., Sahoo,R.K., Sarangi, S.K**. Mathematical Modellling of the working cycle of oil injected Rotary Twin-screw compressor,Applied Thermal Engineering .

[31] www.city.ac.uk/

[32] www.cfd-online.com